Efficient Simulation of Flow and Heat Transfer in Arbitrarily Shaped Pipes
Copyright ©2012 by Patricio I. Rosen Esquivel, Eindhoven, The Netherlands. All rights are reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form or by any means, electronic, mechanical, photocopying, recording or otherwise, without prior permission of the author.

Cover shows a photo of the Trans-Alaska pipeline system (Copyright by Luca Galuzzi)

Cover design by Jorge Israel Garza Burela

A catalogue record is available from the Eindhoven University of Technology Library


NUR: 919

The work presented in this thesis was carried out under the auspices of

Nieuwegein, The Netherlands.
Efficient Simulation of Flow and Heat Transfer in Arbitrarily Shaped Pipes

PROEFSCHRIFT

ter verkrijging van de graad van doctor aan de Technische Universiteit Eindhoven, op gezag van de rector magnificus, prof.dr.ir. C.J. van Duijn, voor een commissie aangewezen door het College voor Promoties in het openbaar te verdedigen op woensdag 21 november 2012 om 16.00 uur

door

Patricio Ivan Rosen Esquivel

geboren te Mexico-Stad, Mexico
Dit proefschrift is goedgekeurd door de promotoren:

prof.dr. R.M.M. Mattheij
en
prof.dr.ir. J.A.M. Dam

Copromotor:
dr.ir. J.H.M. ten Thije Boonkkamp
Nomenclature

Dimensionless Numbers

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Re&lt;sub&gt;est&lt;/sub&gt;</td>
<td>estimation of the local Reynolds number, page 161</td>
</tr>
<tr>
<td>Re&lt;sub&gt;loc&lt;/sub&gt;</td>
<td>local Reynolds number, page 161</td>
</tr>
<tr>
<td>Br</td>
<td>Brinkman number, page 20</td>
</tr>
<tr>
<td>Ec</td>
<td>Eckert number, page 21</td>
</tr>
<tr>
<td>Gr</td>
<td>Grashof number, page 20</td>
</tr>
<tr>
<td>Gr&lt;sub&gt;mod&lt;/sub&gt;</td>
<td>modified Grashof number, page 87</td>
</tr>
<tr>
<td>Pe</td>
<td>Péclet number, page 21</td>
</tr>
<tr>
<td>Pr</td>
<td>Prandtl number, page 20</td>
</tr>
<tr>
<td>Ra</td>
<td>Rayleigh number, page 21</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number, page 20</td>
</tr>
<tr>
<td>Nu&lt;sub&gt;D&lt;/sub&gt;</td>
<td>Nusselt number based on the diameter, page 101</td>
</tr>
</tbody>
</table>

General

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>tensor, page 12</td>
</tr>
<tr>
<td>α</td>
<td>vector, page 12</td>
</tr>
<tr>
<td>e&lt;sub&gt;1&lt;/sub&gt;, e&lt;sub&gt;2&lt;/sub&gt;, e&lt;sub&gt;3&lt;/sub&gt;</td>
<td>cartesian basis vectors, page 182</td>
</tr>
<tr>
<td>e&lt;sub&gt;r&lt;/sub&gt;, e&lt;sub&gt;φ&lt;/sub&gt;, e&lt;sub&gt;x&lt;/sub&gt;</td>
<td>cylindrical basis vectors, page 182</td>
</tr>
<tr>
<td>r, φ, x</td>
<td>cylindrical coordinates [m], page 181</td>
</tr>
<tr>
<td>a</td>
<td>scalar, page 12</td>
</tr>
<tr>
<td>A, A&lt;sub&gt;0&lt;/sub&gt;</td>
<td>constant, page 12</td>
</tr>
<tr>
<td>x&lt;sub&gt;1&lt;/sub&gt;, x&lt;sub&gt;2&lt;/sub&gt;, x&lt;sub&gt;3&lt;/sub&gt;</td>
<td>cartesian coordinates [m], page 181</td>
</tr>
</tbody>
</table>

Greek

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>α&lt;sub&gt;V&lt;/sub&gt;</td>
<td>volumetric thermal expansion coefficient [K&lt;sup&gt;-1&lt;/sup&gt;], page 16</td>
</tr>
<tr>
<td>α&lt;sub&gt;in&lt;/sub&gt;</td>
<td>kinetic energy correction factor at inlet, page 32</td>
</tr>
<tr>
<td>α&lt;sub&gt;out&lt;/sub&gt;</td>
<td>kinetic energy correction factor at outlet, page 32</td>
</tr>
<tr>
<td>α&lt;sub&gt;w&lt;/sub&gt;</td>
<td>wall heat transfer coefficient [W m&lt;sup&gt;-2&lt;/sup&gt; K&lt;sup&gt;-1&lt;/sup&gt;], page 86</td>
</tr>
<tr>
<td>β</td>
<td>average pressure gradient in axial direction [Pa m&lt;sup&gt;-1&lt;/sup&gt;], page 46</td>
</tr>
<tr>
<td>γ</td>
<td>average axial temperature gradient [K m&lt;sup&gt;-1&lt;/sup&gt;], page 78</td>
</tr>
<tr>
<td>δ</td>
<td>nearest distance to the wall [m], page 160</td>
</tr>
<tr>
<td>η</td>
<td>radial boundary layer variable, page 103</td>
</tr>
<tr>
<td>ε</td>
<td>small parameter, page 53</td>
</tr>
<tr>
<td>ε&lt;sub&gt;ε&lt;/sub&gt;</td>
<td>specific dissipation rate of turbulent kinetic energy [J kg&lt;sup&gt;-1&lt;/sup&gt; s&lt;sup&gt;-1&lt;/sup&gt;], page 150</td>
</tr>
<tr>
<td>Γ</td>
<td>wall of the pipe, page 30</td>
</tr>
<tr>
<td>Γ&lt;sub&gt;0&lt;/sub&gt;</td>
<td>homogenized wall, page 101</td>
</tr>
<tr>
<td>Γ&lt;sub&gt;c&lt;/sub&gt;</td>
<td>corrugated wall, page 100</td>
</tr>
<tr>
<td>Γ&lt;sub&gt;E&lt;/sub&gt;</td>
<td>center line, page 100</td>
</tr>
<tr>
<td>Γ&lt;sub&gt;in&lt;/sub&gt;</td>
<td>inlet boundary, page 28</td>
</tr>
<tr>
<td>Γ&lt;sub&gt;out&lt;/sub&gt;</td>
<td>outlet boundary, page 28</td>
</tr>
<tr>
<td>κ</td>
<td>coefficient of heat conduction [W m&lt;sup&gt;-1&lt;/sup&gt; K&lt;sup&gt;-1&lt;/sup&gt;], page 14</td>
</tr>
<tr>
<td>λ</td>
<td>second viscosity [Pa s], page 14</td>
</tr>
<tr>
<td>μ</td>
<td>dynamic viscosity [Pa s], page 14</td>
</tr>
<tr>
<td>μ&lt;sub&gt;r&lt;/sub&gt;</td>
<td>effective dynamic viscosity [Pa s], page 150</td>
</tr>
<tr>
<td>μ&lt;sub&gt;t&lt;/sub&gt;</td>
<td>turbulent dynamic viscosity [Pa s], page 149</td>
</tr>
<tr>
<td>ν</td>
<td>kinematic viscosity [m&lt;sup&gt;2&lt;/sup&gt; s&lt;sup&gt;-1&lt;/sup&gt;], page 18</td>
</tr>
<tr>
<td>Ω</td>
<td>fluid domain, page 28</td>
</tr>
<tr>
<td>Ω&lt;sub&gt;0&lt;/sub&gt;</td>
<td>homogenized domain, page 101</td>
</tr>
<tr>
<td>Ω&lt;sub&gt;c&lt;/sub&gt;</td>
<td>domain of corrugated pipe, page 98</td>
</tr>
<tr>
<td>Ψ</td>
<td>heat source [W m&lt;sup&gt;-3&lt;/sup&gt;], page 12</td>
</tr>
</tbody>
</table>

T | stress tensor [Pa], page 12 |
<p>| Δp | pressure drop [Pa], page 26 |
| Δp&lt;sub&gt;S&lt;/sub&gt; | pressure drop due to pressure drop due to skin friction [Pa], page 31 |
| Δp&lt;sub&gt;p&lt;/sub&gt; | pressure drop due to pressure forces at the wall [Pa], page 31 |
| Δρ | typical density difference [kg m&lt;sup&gt;-3&lt;/sup&gt;], page 17 |
| ΔT | typical temperature difference [K], page 17 |
| δ | nearest distance to the wall [m], page 160 |
| η | radial boundary layer variable, page 103 |
| ε | small parameter, page 53 |
| ε&lt;sub&gt;ε&lt;/sub&gt; | specific dissipation rate of turbulent kinetic energy [J kg&lt;sup&gt;-1&lt;/sup&gt; s&lt;sup&gt;-1&lt;/sup&gt;], page 150 |</p>
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \rho )</td>
<td>fluid density ([\text{kg m}^{-3}])</td>
<td>12</td>
</tr>
<tr>
<td>( \tau_w )</td>
<td>wall shear stress [Pa]</td>
<td>154</td>
</tr>
<tr>
<td>( \varphi_\Omega )</td>
<td>specific dissipation within ( \Omega ), [J kg(^{-1})]</td>
<td>32</td>
</tr>
<tr>
<td>( \xi )</td>
<td>fast axial variable, page 103</td>
<td></td>
</tr>
<tr>
<td>( \tau )</td>
<td>viscous stress tensor [Pa], page 13</td>
<td></td>
</tr>
<tr>
<td>( q )</td>
<td>heat flux [W m(^{-2})], page 12</td>
<td></td>
</tr>
<tr>
<td>( \dot{m} )</td>
<td>mass flow rate [kg s(^{-1})]</td>
<td>32</td>
</tr>
<tr>
<td>( \dot{W}_{\text{pump}} )</td>
<td>pump effective power [W], page 32</td>
<td></td>
</tr>
<tr>
<td>( K )</td>
<td>von Kármán constant, page 155</td>
<td></td>
</tr>
<tr>
<td>( R )</td>
<td>specific gas constant [J kg(^{-1}) K(^{-1})], page 14</td>
<td></td>
</tr>
<tr>
<td>( p )</td>
<td>pressure [Pa], page 13</td>
<td></td>
</tr>
<tr>
<td>( p_t )</td>
<td>turbulent pressure [Pa], page 150</td>
<td></td>
</tr>
<tr>
<td>( R_{w(x)} )</td>
<td>pipe radius function [m], page 28</td>
<td></td>
</tr>
<tr>
<td>( R_{\text{w}}^0 )</td>
<td>minimum pipe radius, page 100</td>
<td></td>
</tr>
<tr>
<td>( T )</td>
<td>temperature [K], page 13</td>
<td></td>
</tr>
<tr>
<td>( t )</td>
<td>time [s], page 12</td>
<td></td>
</tr>
<tr>
<td>( T_e )</td>
<td>environmental temperature [K], page 84</td>
<td></td>
</tr>
<tr>
<td>( u )</td>
<td>flow velocity [m s(^{-1})], page 12</td>
<td></td>
</tr>
<tr>
<td>( u_c )</td>
<td>average parallel velocity component [m s(^{-1})], page 154</td>
<td></td>
</tr>
<tr>
<td>( u_f )</td>
<td>friction velocity [m s(^{-1})], page 154</td>
<td></td>
</tr>
<tr>
<td>( y_p )</td>
<td>distance at which the boundary conditions are imposed for the ( k-\varepsilon ) model [m], page 155</td>
<td></td>
</tr>
<tr>
<td>( a )</td>
<td>amplitude of corrugations [m], page 49</td>
<td></td>
</tr>
<tr>
<td>( C )</td>
<td>constant from logarithmic law of the wall, page 155</td>
<td></td>
</tr>
<tr>
<td>( C_V )</td>
<td>specific heat at constant volume [J kg(^{-1}) K(^{-1})], page 15</td>
<td></td>
</tr>
<tr>
<td>( C_p )</td>
<td>specific heat at constant pressure [J kg(^{-1}) K(^{-1})], page 15</td>
<td></td>
</tr>
<tr>
<td>( CF_1 )</td>
<td>friction correction factor, page 57</td>
<td></td>
</tr>
<tr>
<td>( CF_2 )</td>
<td>improved friction correction factor, page 59</td>
<td></td>
</tr>
<tr>
<td>( n )</td>
<td>outer unit normal vector, page 25</td>
<td></td>
</tr>
<tr>
<td>( D )</td>
<td>pipe diameter [m], page 26</td>
<td></td>
</tr>
<tr>
<td>( E )</td>
<td>expansion length [m], page 63</td>
<td></td>
</tr>
<tr>
<td>( e )</td>
<td>specific internal energy [J kg(^{-1})], page 12</td>
<td></td>
</tr>
<tr>
<td>( Err )</td>
<td>relative error, page 60</td>
<td></td>
</tr>
<tr>
<td>( f )</td>
<td>Darcy friction factor, page 26</td>
<td></td>
</tr>
<tr>
<td>( H )</td>
<td>Total heat transfer [W], page 87</td>
<td></td>
</tr>
<tr>
<td>( \dot{m} )</td>
<td>mass flow rate [kg s(^{-1})]</td>
<td></td>
</tr>
<tr>
<td>( q )</td>
<td>heat flux [W m(^{-2})]</td>
<td></td>
</tr>
<tr>
<td>( \tau_w )</td>
<td>wall shear stress [Pa]</td>
<td></td>
</tr>
<tr>
<td>( \varphi_\Omega )</td>
<td>specific dissipation within ( \Omega ), [J kg(^{-1})]</td>
<td></td>
</tr>
<tr>
<td>( \xi )</td>
<td>fast axial variable, page 103</td>
<td></td>
</tr>
<tr>
<td>( \tau )</td>
<td>viscous stress tensor [Pa], page 13</td>
<td></td>
</tr>
<tr>
<td>( q )</td>
<td>heat flux [W m(^{-2})]</td>
<td></td>
</tr>
<tr>
<td>( \dot{m} )</td>
<td>mass flow rate [kg s(^{-1})]</td>
<td></td>
</tr>
<tr>
<td>( \dot{W}_{\text{pump}} )</td>
<td>pump effective power [W]</td>
<td></td>
</tr>
<tr>
<td>( K )</td>
<td>von Kármán constant</td>
<td></td>
</tr>
<tr>
<td>( R )</td>
<td>specific gas constant [J kg(^{-1}) K(^{-1})]</td>
<td></td>
</tr>
<tr>
<td>( p )</td>
<td>pressure [Pa]</td>
<td></td>
</tr>
<tr>
<td>( p_t )</td>
<td>turbulent pressure [Pa]</td>
<td></td>
</tr>
<tr>
<td>( R_{w(x)} )</td>
<td>pipe radius function [m]</td>
<td></td>
</tr>
<tr>
<td>( R_{\text{w}}^0 )</td>
<td>minimum pipe radius</td>
<td></td>
</tr>
<tr>
<td>( T )</td>
<td>temperature [K]</td>
<td></td>
</tr>
<tr>
<td>( t )</td>
<td>time [s]</td>
<td></td>
</tr>
<tr>
<td>( T_e )</td>
<td>environmental temperature [K]</td>
<td></td>
</tr>
<tr>
<td>( u )</td>
<td>flow velocity [m s(^{-1})]</td>
<td></td>
</tr>
<tr>
<td>( u_c )</td>
<td>average parallel velocity component [m s(^{-1})]</td>
<td></td>
</tr>
<tr>
<td>( u_f )</td>
<td>friction velocity [m s(^{-1})]</td>
<td></td>
</tr>
<tr>
<td>( y_p )</td>
<td>distance at which boundary conditions are imposed for the ( k-\varepsilon ) model [m]</td>
<td></td>
</tr>
<tr>
<td>( a )</td>
<td>amplitude of corrugations [m]</td>
<td></td>
</tr>
<tr>
<td>( C )</td>
<td>constant from logarithmic law of the wall</td>
<td></td>
</tr>
<tr>
<td>( C_V )</td>
<td>specific heat at constant volume [J kg(^{-1}) K(^{-1})]</td>
<td></td>
</tr>
<tr>
<td>( C_p )</td>
<td>specific heat at constant pressure [J kg(^{-1}) K(^{-1})]</td>
<td></td>
</tr>
<tr>
<td>( CF_1 )</td>
<td>friction correction factor</td>
<td></td>
</tr>
<tr>
<td>( CF_2 )</td>
<td>improved friction correction factor</td>
<td></td>
</tr>
<tr>
<td>( n )</td>
<td>outer unit normal vector</td>
<td></td>
</tr>
<tr>
<td>( D )</td>
<td>pipe diameter [m]</td>
<td></td>
</tr>
<tr>
<td>( E )</td>
<td>expansion length [m]</td>
<td></td>
</tr>
<tr>
<td>( e )</td>
<td>specific internal energy [J kg(^{-1})]</td>
<td></td>
</tr>
<tr>
<td>( Err )</td>
<td>relative error</td>
<td></td>
</tr>
<tr>
<td>( f )</td>
<td>Darcy friction factor</td>
<td></td>
</tr>
<tr>
<td>( H )</td>
<td>Total heat transfer [W]</td>
<td></td>
</tr>
<tr>
<td>Symbol</td>
<td>Definition</td>
<td>Page(s)</td>
</tr>
<tr>
<td>--------</td>
<td>---------------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>$h$</td>
<td>enthalpy [J kg$^{-1}$], page 13</td>
<td></td>
</tr>
<tr>
<td>$k$</td>
<td>specific turbulent kinetic energy [J kg$^{-1}$], page 148</td>
<td></td>
</tr>
<tr>
<td>$K_e$</td>
<td>loss coefficient, page 32</td>
<td></td>
</tr>
<tr>
<td>$L$</td>
<td>pipe period [m], page 30</td>
<td></td>
</tr>
<tr>
<td>$L_e$</td>
<td>ratio of expansion length and period $L_e = E/L$, page 64</td>
<td></td>
</tr>
<tr>
<td>$L_p$</td>
<td>total pipe length [m], page 81</td>
<td></td>
</tr>
<tr>
<td>$p_h$</td>
<td>hydrostatic pressure [Pa], page 18</td>
<td></td>
</tr>
<tr>
<td>$Q$</td>
<td>volumetric flow rate $[m^3 s^{-1}]$, page 54</td>
<td></td>
</tr>
<tr>
<td>$Q_{max}$</td>
<td>maximum flow rate $[m^3 s^{-1}]$, page 66</td>
<td></td>
</tr>
<tr>
<td>$q_w$</td>
<td>wall heat flux $[W m^{-2}]$, page 75</td>
<td></td>
</tr>
<tr>
<td>$s$</td>
<td>entropy [J kg$^{-1}$ K$^{-1}$], page 13</td>
<td></td>
</tr>
<tr>
<td>$s_{irr}$</td>
<td>irreversible entropy, page 37</td>
<td></td>
</tr>
<tr>
<td>$s_{rev}$</td>
<td>reversible entropy, page 37</td>
<td></td>
</tr>
<tr>
<td>$T_{max}$</td>
<td>maximum temperature in the domain [K], page 87</td>
<td></td>
</tr>
<tr>
<td>$U_x$</td>
<td>axial average velocity $[m s^{-1}]$, page 29</td>
<td></td>
</tr>
</tbody>
</table>

**Subscripts**

- $()_0$ typical scale, page 17
- $()_{\phi}$ azimuthal component, page 29
- $()_r$ radial component, page 29
- $()_x$ axial component, page 29
- $()_{in}$ at inlet, page 32
- $()_{out}$ at outlet, page 32
- $()_{ref}$ reference value, page 16
- $()_{sp}$ straight pipe, page 44

**Overscripts**

- $\tilde{}$ slowly varying variable, page 53
- $\bar{}$ time averaged variable, page 144
- $\hat{}$ fluctuation due to the corrugation, page 46
# Contents

1 Introduction .................................................. 1
   1.1 Motivation .................................................. 1
   1.1.1 Pipeline Transport ..................................... 1
   1.1.2 Corrugated Pipes ...................................... 2
   1.2 Problem description ...................................... 4
   1.3 Objective of this Thesis .................................. 6
   1.4 Thesis Outline ............................................ 8

2 Governing Equations for Fluid Flow and Heat Transfer .............. 11
   2.1 Fluid Mechanics ............................................ 11
   2.2 Conservation Laws ........................................ 12
   2.2.1 Stress Decomposition .................................. 12
   2.2.2 Entropy Equation ....................................... 13
   2.2.3 Constitutive Equations ................................ 14
   2.3 Oberbeck-Boussinesq Approximation ....................... 15
   2.3.1 Thermal Expansion and Compressibility ............... 16
   2.3.2 Continuity Equation ................................... 17
   2.3.3 Momentum Equations ................................... 18
   2.3.4 Temperature Equation .................................. 19
   2.4 Dimensional Analysis ...................................... 20
   2.4.1 Forced Convection ..................................... 22
   2.4.2 Natural Convection ..................................... 23
   2.4.3 Boundary Conditions for Fluid Flow .................. 24
   2.5 Flow in Pipes .............................................. 26
   2.5.1 The Darcy Friction Factor ............................ 26
   2.5.2 Corrugated Pipes ...................................... 27
   2.5.3 Integral Expression for the Friction Factor ........... 29
   2.5.4 Periodic Pipes ......................................... 30
   2.6 Flow in Arbitrary Conduits ................................ 31
   2.6.1 Loss Coefficients ..................................... 31
   2.6.2 Bernoulli’s Equation ................................. 33
   2.6.3 Entropy Production ................................... 36


3 Isothermal Laminar Flow

3.1 Introduction ................................................. 40
3.2 Governing Equations and Geometry .............................. 41
3.2.1 The Darcy Friction Factor ................................. 42
3.3 Numerical Model .............................................. 45
3.3.1 Mesh Independence ....................................... 48
3.3.2 Validation ................................................ 49
3.4 An Asymptotic Formula for the Friction Factor ................. 51
3.4.1 Method of Slow Variations ................................ 52
3.4.2 Reformulation in Slowly Varying Variables ............... 53
3.4.3 Leading Order Solution .................................. 55
3.4.4 Estimation of the Friction Factor ......................... 56
3.4.5 Experimental and Numerical Validation .................... 60
3.4.6 Applicability of the Method ............................... 61
3.5 Wall-Shape Design ............................................. 63
3.5.1 The Cavity Configuration .................................. 64
3.5.2 The Slowly Varying Configuration ......................... 68
3.5.3 Remarks on Wall-Shape Design ......................... 70

4 Non-Isothermal Laminar Flow ................................. 73
4.1 Convective Heat Transfer ..................................... 74
4.2 Forced Convection ............................................ 74
4.2.1 Uniform Heat Flux Problem ................................ 75
4.3 Solution with Periodicity Decomposition ....................... 76
4.3.1 Flow Equations .......................................... 76
4.3.2 Temperature Equation .................................... 77
4.3.3 Evaluation of Temperature Gradient ....................... 78
4.4 Natural Convection ............................................ 79
4.4.1 Natural Convection Loops ................................ 80
4.4.2 Material Properties ...................................... 82
4.5 Modeling of a Thermosyphon Loop ............................ 84
4.5.1 Geometry and Governing Equations ....................... 85
4.5.2 Boundary Conditions ..................................... 86
4.5.3 Numerical Methodology .................................. 87
4.6 Effects of Wall-Shape ........................................ 89
4.6.1 Influence of Corrugation Size $a$ ......................... 89
4.6.2 Influence of Proportion of Expansion $L_e$ ............... 91
4.7 Remarks on Non-Isothermal Laminar Flow .................... 94

5 Homogenization of the Boussinesq Equations .................. 97
5.1 Idea of Homogenization ...................................... 97
5.2 Mathematical Model ......................................... 98
5.2.1 Natural Convection Equations ............................ 98
5.2.2 Geometry and Boundary Conditions ....................... 99
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.3 Formal Homogenization</td>
<td>101</td>
</tr>
<tr>
<td>5.3.1 Asymptotic Methods</td>
<td>102</td>
</tr>
<tr>
<td>5.3.2 Differentiation Rules</td>
<td>104</td>
</tr>
<tr>
<td>5.4 Homogenization of the Temperature Equation</td>
<td>105</td>
</tr>
<tr>
<td>5.4.1 Outer Equation</td>
<td>105</td>
</tr>
<tr>
<td>5.4.2 Inner Equation</td>
<td>106</td>
</tr>
<tr>
<td>5.4.3 Boundary Conditions</td>
<td>107</td>
</tr>
<tr>
<td>5.4.4 Matching Conditions</td>
<td>108</td>
</tr>
<tr>
<td>5.4.5 Effective Boundary Condition</td>
<td>111</td>
</tr>
<tr>
<td>5.5 Homogenization of the Flow Equations</td>
<td>115</td>
</tr>
<tr>
<td>5.5.1 Outer Solution</td>
<td>115</td>
</tr>
<tr>
<td>5.5.2 Homogenized Equations</td>
<td>117</td>
</tr>
<tr>
<td>5.5.3 Inner Solution</td>
<td>118</td>
</tr>
<tr>
<td>5.5.4 Effective Boundary Condition</td>
<td>119</td>
</tr>
<tr>
<td>5.6 Homogenized Boussinesq System of Equations</td>
<td>120</td>
</tr>
<tr>
<td>5.7 Numerical Analysis</td>
<td>121</td>
</tr>
<tr>
<td>5.7.1 Mesh Generation</td>
<td>121</td>
</tr>
<tr>
<td>5.7.2 A Geometry with Small $\epsilon$</td>
<td>125</td>
</tr>
<tr>
<td>5.7.3 A Geometry with Moderate $\epsilon$</td>
<td>131</td>
</tr>
<tr>
<td>5.7.4 Accuracy of the Method for Large Wall Heat Transfer Coefficients</td>
<td>135</td>
</tr>
<tr>
<td>5.7.5 Speed Up of the Homogenization Approach</td>
<td>138</td>
</tr>
<tr>
<td>6 Isothermal Turbulent Flow</td>
<td>141</td>
</tr>
<tr>
<td>6.1 Turbulence Modeling</td>
<td>141</td>
</tr>
<tr>
<td>6.1.1 Reynolds Averaged Navier-Stokes Equations (RANS)</td>
<td>142</td>
</tr>
<tr>
<td>6.1.2 Properties of Time Averaging</td>
<td>143</td>
</tr>
<tr>
<td>6.1.3 Derivation of the RANS Equations</td>
<td>143</td>
</tr>
<tr>
<td>6.1.4 Time-Dependent RANS</td>
<td>146</td>
</tr>
<tr>
<td>6.1.5 Turbulence Closure Models</td>
<td>147</td>
</tr>
<tr>
<td>6.1.6 Boussinesq Turbulent-Viscosity Assumption</td>
<td>147</td>
</tr>
<tr>
<td>6.2 The $k$-$\epsilon$ Model</td>
<td>148</td>
</tr>
<tr>
<td>6.2.1 Transport Equations for $k$ and $\epsilon$</td>
<td>149</td>
</tr>
<tr>
<td>6.2.2 Wall-Functions</td>
<td>150</td>
</tr>
<tr>
<td>6.2.3 Boundary Conditions for the $k$-$\epsilon$ Model</td>
<td>153</td>
</tr>
<tr>
<td>6.3 The LVEL Model</td>
<td>156</td>
</tr>
<tr>
<td>6.3.1 Spalding’s Law of The Wall</td>
<td>156</td>
</tr>
<tr>
<td>6.3.2 Turbulent Viscosity and Local Reynolds Number</td>
<td>157</td>
</tr>
<tr>
<td>6.3.3 Solution Algorithm</td>
<td>159</td>
</tr>
<tr>
<td>6.4 Setup for Periodic Corrugated Pipes</td>
<td>161</td>
</tr>
<tr>
<td>6.4.1 Periodicity Decomposition</td>
<td>162</td>
</tr>
<tr>
<td>6.4.2 The $k$-$\epsilon$ System of Equations</td>
<td>163</td>
</tr>
<tr>
<td>6.4.3 The LVEL System of Equations</td>
<td>163</td>
</tr>
<tr>
<td>6.5 Numerical Analysis</td>
<td>164</td>
</tr>
<tr>
<td>6.5.1 Discretization</td>
<td>164</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

1.1 Motivation

Historically, the theories of fluid dynamics and heat transfer were developed independently as branches of physics. Fluid dynamics is concerned with the flow of fluids and heat transfer with the exchange of energy. The unified study of these phenomena is crucial in engineering sciences and they have evolved into a unified theory under the name of transport phenomena. The development of this unified discipline continues to grow and it finds applications in a very broad variety of fields such as biotechnology, microelectronics, cryogenic engineering, chemical engineering, meteorology, etc. [8]. Problems involving flow and heat transfer occur also at all scales, ranging from microscopic biological systems to automobiles, airplanes, spacecraft propulsion and large scale geophysical flows, etc. [15, 55].

1.1.1 Pipeline Transport

The problem that is the main subject of this thesis is related to the commonly encountered application of transporting fluids through pipes or conduits. This is one of the very first engineering problems humankind faced as cities were developing and therefore increasing their demand of water supply, for domestic and agricultural use. Every successful civilization of ancient times invested considerable efforts in the construction and the maintenance of water systems. The Romans are famous for their aqueducts, like the 50 km long Nîmes aqueduct, built around 50 BC [13]. The “Pont du Gard” bridge which crosses the Gardon river in southern France, is part of this ancient aqueduct, see Figure
1.1(a). In the pre-Columbian era, the capital of the Aztec empire “Tenochtitlan”, located in the shallow lake “Texcoco” (see the center of Figure 1.1(b)), featured causeways which connected the city in the middle of the lake with the mainland. Along these causeways they installed two double pipe aqueducts made of terracotta of about 4 km each. These aqueducts provided the city with fresh water from the springs at Chapultepec (located in middle left side of Figure 1.1(b)) [85].

Hydraulic engineering continue to develop through time. During the islamic golden age, between the 8th and 16th centuries, water management technologies were assembled, standardized and spread to the rest of the old world. These technologies featured canals, pipes, dams, and water lifting devices like screwpumps and norias, among others [14]. Many of the ancient technologies are still in use, liquids are still moved by gravity through aqueducts and pipes, although now many other modern devices are used. Modern versions of screwpumps for example, are used for draining the Polders of Kinderdijk-Elshout in the Netherlands, replacing the function of its world famous dutch windmills. The windmills are maintained in operating conditions and they can still be used as fall-back in case of failure of the modern equipment [37]. Nowadays, the transport of fluids through pipelines is a very commonly encountered industrial practice. Oil and natural gas are transported hundreds or even thousands of kilometers along pipelines. The Trans-Alaska pipeline system built between 1974 and 1977 (see Figure 1.2) for example, is a 1288 km long pipeline which transports oil from Prudhoe Bay to Valdez Alaska [20]. The coolant in an engine is transported through conduits from the engine to the radiator for cooling the engine and complex piping systems are used for conveying fluids between the different operating units in process and energy plants for example.

1.1.2 Corrugated Pipes

Despite their antiquity, there are still research and developments on the design of pipes and piping related technologies. Some properties, such as portability and flexibility, are convenient characteristics of the so-called corrugated pipes. These are pipes with non-straight walls, i.e., with corrugations. Figures 1.3(a) and 1.3(b) show an outer and internal view of a typical periodic corrugated pipe, respectively. The corrugations are responsible for the enhancement of the local stiffness of the pipe, but at the same time, they also provide flexibility on the long scale. These properties allow the pipes to be bended without collapsing and it makes them convenient from an operational point of view. For example, in Chapter 4 of this thesis, we treat the case of a cryogenic storage tank featuring a corrugated thermosyphon return line. In this particular application, the corrugated pipe provides very convenient installation and maintenance possibilities. For instance, if the delivery pump needs to be fixed or replaced, one can simply close the valve and detach the corrugated pipe, exploiting the flexibility and hence in-
1.1 Motivation

Figure 1.1: Figure 1.1(a) displays a photograph of the “Pont du Gard”, bridge of a Roman aqueduct which has been included in the World Heritage List; photo courtesy of UNESCO World Heritage [38]. Figure 1.1(b) shows an artistic representation of the capital of the Aztec Empire “Tenochtitlan” located in the middle of the Texcoco lake (currently Mexico City). Along the causeways, two double aqueducts of more than 4 km each, provided the city with water from the springs at Chapultepec (located in the middle left side of the image); the image is courtesy of the artist Tomás J. Filsinger [29].

These properties, among others, make corrugated pipes suitable for many different applications, including domestic appliances, heating, ventilation, air-conditioning, cryogenic facilities, and the LNG (Liquefied Natural Gas) industry, for example.

In the particular case of the LNG industry, the flexibility of corrugated pipes has made it possible to develop innovative offshore transfer systems. Usually, LNG was transferred between ships and onshore facilities by the use of fixed articulation loading arms. These arms included swivel joints which allowed to correct the relative motion between the onshore fixed station and the offshore buoyant ship. However, such a transfer system can only be used under mild conditions. The situation becomes more difficult in applications involving two moving objects, as for instance in a ship to ship transfer. For these kinds of applications, flexibility is crucial in order to avoid the danger of rupture. Corrugated pipes have been used for designing transfer and offloading systems which can operate under more diverse conditions [116]. Corrugated pipes are also used as flexible rises, these are pipes that allow fluid flow between a sea installation and a facility on a drill platform, again the flexibility is a key aspect [90]. However, corrugated pipes are not restricted to these type of applications and they also find applications in the design of heat exchangers, were they are used with the purpose of enhancing the heat transfer [24].
1.2 Problem description

Corrugated pipes have certain characteristics that make them appealing from an operational point of view, such as flexibility and local stiffness. These properties are directly related to the corrugation pattern with which these type of pipes are designed. However, at the same time, the introduction of the corrugated walls poses a fundamental question, namely, how do the corrugations affect the mass, momentum and energy transfer inside the pipe?

This question is quite general and it gives rise to the study of many types of effects. Some studies concentrate on acoustic effects, like the self-sustained oscillations induced by the corrugations [74], or on mechanical properties of the pipes themselves [128]. In this thesis we concentrate on the effects created by the wall-shape on the flow and heat transfer. We put particular attention to the impact of the wall-shape on the losses of
mechanical energy, which are commonly referred to as pressure losses. These losses are a key aspect in engineering applications, because they determine the power requirements for maintaining the desired flow rate. We also consider heat transfer problems in corrugated pipes in the limiting cases of forced and natural convection.

Besides the diversity of effects that one can study, there are also different approaches that one can take in order to study the impact of wall-shape. Historically, theory and experimentation developed together and have helped in understanding a large variety of physical phenomena. The advent of fast computers and the equally fast development of numerical methods has allowed to simulate physical phenomena with the aid of a computer. This has given rise to computational science, which has become an important area of science along with theory and experiment [80]. Fluid mechanics and transport phenomena problems, are no exception and they require theory, experimentation as well as numerical simulation. Numerical simulation for fluid dynamics has become a field on its own under the name of Computational Fluid Dynamics (CFD).

Physical experimentation has the great advantage of retrieving information directly from measurements; however, it can easily become very expensive and unpractical for certain tasks, such as for optimization or in the design of a large network of interconnected hydraulic components. Theory does not only provide the basic modeling equations on which numerical methods are based, but also allows the usage of analytical approaches for the development of new models and for deriving exact or approximate analytical solutions. Nonetheless, analytical solutions are difficult to find and can only be obtained for certain specific problems. The steady developments in CFD, have broaden the variety of engineering problems which can be tackled with the aid of a computer. However, this does not mean that it is possible to take any kind of problem and simply put it into a computer simulation program to obtain the answer, as there are several restrictions and challenges.

One of the most important challenges is related to the complexity of flows at high Reynolds numbers, i.e., turbulent flows. In order to make an accurate numerical simulation of a turbulent flow, all relevant scales must be solved. However, the ratio of smallest to largest scales decreases rapidly as the Reynolds number increases, this strongly restricts the usage of direct numerical simulation to moderate Reynolds numbers [71, 126]. A second very important challenge, on which the present thesis is focused, is related to the simulation of flows and heat transfer through complex and/or extended geometries, for both laminar and turbulent flow conditions. A complex boundary, such as the wall of a corrugated pipe for instance, can introduce extra length scales to the problem and significantly increase the difficulty of finding a solution. Even in the case of laminar flow, a complex geometry can represent a big challenge. Problems with extended geometries, i.e., problems in which the computational domain can not be simplified by periodicity or symmetry argument, also have a great impact on the difficulty of the problem, specially when combined with a geometry with multiple scales.
The previous issues can make the computational time too restrictive, specially for engineering applications which require computing a solution for several geometries. Therefore there is a strong need for simplified models and more efficient numerical methods, which can handle the simulation of flow and heat transfer in complex and extended geometries. The present thesis is oriented to contribute in this direction. With this goal in mind we consider an approach which exploits both, analytical and numerical techniques.

1.3 Objective of this Thesis

In short, the main objective of this thesis is the development of simplified models and efficient numerical methods for simulating fluid flow and heat transfer in arbitrarily shaped pipes.

In order to pursue this goal, we subdivide the study in the thesis into three main categories, namely, isothermal laminar flow, non-isothermal laminar flow and isothermal turbulent flow. From the basic laws of conservation of mass, momentum and energy, we derive the Boussinesq approximation, which is based on the assumption of quasi-incompressibility. By using dimensional analysis, we present the limiting cases of forced and natural convection. This in turn, provides us with the equations required for modeling the physical phenomena involved.

Due to their importance in industrial applications, we pay special attention to the computation of the losses of mechanical energy. In practice they are expressed in terms of friction factors or loss coefficients. Starting from the governing equations, we obtain integral expressions for the friction factor and loss coefficients. We also obtain a generalization of the Bernoulli’s equation which allows us to describe the losses of mechanical energy in an arbitrarily shaped three-dimensional conduit. In parallel, this derivation allows us to establish a connection between the losses of mechanical energy and the production of entropy in the flow.

The governing equations and formulas developed in the first part of the thesis form the basis for the subsequent results. In the case of isothermal laminar flow, we combine the integral expression for the friction factor in periodic pipes with the asymptotic method of slow variations for deriving an analytical formula for the friction factor. The developed analytical formula is very efficient because it only requires numerical integration in one dimension, when compared to a two-dimensional finite element numerical model, the analytical formula is about 1000 times faster.

We show that the friction factor can be significantly affected by the corrugations. This is in contrast to the Moody diagram, which shows the friction factor as independent of
1.3 Objective of this Thesis

roughness for the laminar regime. We also present a finite element model which exploits the periodicity of the flow for reducing the computational efforts. The model allows us to systematically evaluate the accuracy of the analytical formula and it also provides us with an alternative for computing the flow field and the friction factor in the cases when the analytical formula is not accurate. In addition, we also show how our models and numerical methods, can be exploited not only for prediction, but also for wall-shape design and optimization.

In the case of non-isothermal laminar flow, we consider the cases of forced and natural convection. The case of forced convection implies a one-way coupling between the flow equations and the energy equations. We show that this problem can be solved by using a periodicity decomposition, which as in the case of isothermal laminar flow, allows us to reduce the computational domain to just one period. We give extra attention to the case of natural convection, due to the challenges related to the simulation of these type of flows. Due to the two-way coupling between the flow and energy equations, the equations need to be solved simultaneously.

We concentrate first on an industrial application involving a cryogenic storage tank featuring a thermosyphon loop. We present a numerical model to handle this problem; however, in this case, it is no longer possible to reduce the domain by using a periodicity type argument and we are forced to consider the full domain. On the basis of the numerical model, we show that is possible to achieve a better performance by designing the wall-shape. However, the computational costs of such a numerical model are considerable. The main difficulty comes from the appearance of multiple scales, one scale is related to the development of boundary layers along the wall, and the other is related to the small oscillations induced by the presence of the corrugations. In order to capture the multiple scales, one requires a very fine mesh, which in turn makes the application of a direct numerical model quite restrictive. We provide a very convenient alternative to the problem of multiple scales, by using homogenization. The global idea of the approach, is to simplify the domain by replacing the corrugated wall with a flat boundary with certain effective boundary conditions. The key aspect in this approach, is the derivation of the effective boundary conditions, because they are the ones that guarantee that the solution computed on the simplified domain, i.e., the homogenized solution, closely approximates the original solution in the simplified domain. The biggest advantage of the approach, is that one does not need to numerically solve for the small scales, but still, the wall-shape effects are taken into account via the effective boundary conditions. The obtained homogenized model is shown to accurately predict both, local and averaged quantities in the simplified domain in a fraction of the costs of the direct numerical approach.

In the case of isothermal turbulent flow, we start the discussion by introducing the (RANS) Reynolds averaged Navier-Stokes equations, which are the basis for the turbulence models which we use to tackle this problem. We concentrate our attention on
two main models, the two-equation $k-\epsilon$ model and the algebraic type LVEL model. In both models we use periodicity decompositions for reducing the computational costs of the models. In the case of the $k-\epsilon$ model, we first discuss how we need to modify the formula for the friction factor and then we show how to use it for computing the friction factor. As an alternative, we present our implementation of the LVEL turbulence model for computing the friction factor. This alternative proves to be more efficient and robust for handling turbulent flows. We provide comparison with experimental data on corrugated pipes and address the advantages and disadvantages of both methods.

1.4 Thesis Outline

The present thesis is organized in the following way. In Chapter 2, we start by providing a general framework to the governing equations of fluid flow and we present the derivation of models for incompressible flow, forced convection and natural convection. Then we address the modeling of fluid flow in arbitrarily shaped pipes, and discuss the role of friction factors and loss coefficients. Key aspects in this chapter are the mathematical derivations of an integral expression for the friction factor and of a generalization of Bernoulli’s equation which includes the losses of mechanical energy. The integral expression is exploited for obtaining an analytical formula for the friction factor of laminar flows in periodic pipes in Chapter 3. The generalization of Bernoulli’s equation allows us to obtain a representation of the loss coefficients for arbitrary three-dimensional geometries. At the same time, it allows us to establishes a link between the loss coefficients and the entropy production in a flow. This is later exploited in Chapter 6, which is dedicated to turbulence.

Chapters 3, 4 and 5 are devoted to the study of laminar flow, while Chapter 6 is dedicated to the study of turbulent flow. In Chapter 3 we treat the case of isothermal laminar flow. In this chapter we develop a very efficient analytical formula for computing the friction factor by combining the method of slow variations and the integral formula for the friction factor developed in Chapter 2. For non-slowly varying geometries we present an efficient numerical model which exploits periodicity. Based on these models, we address the problem of wall-shape design and show that it is possible to reduce the friction factor. In addition, the range of validity of the models is addressed systematically. Chapter 4 is dedicated to the case of non-isothermal laminar flow. We consider both forced and natural convection. For forced convection, we show that is is possible to exploit the periodicity of the flow and temperature fields in order to reduce the computational cost of implementing a numerical model. In the case of natural convection, we first take a more practically oriented approach and we consider a real-life application involving a thermosyphon loop. We present a numerical model for this problem and show that it is possible to achieve a better design by tuning the wall-shape. At the same time we discuss why in this case it is no longer possible to use a “periodicity type”
decomposition for reducing the computational costs. This issue gives rise to Chapter 5, where we use the method of homogenization in order to simplify the computational geometry and hence significantly reduce the computational costs. The basic idea is to replace the original corrugated wall of the pipe, by a simple straight wall with some special “effective” boundary conditions. The key aspect is that these effective boundary conditions are derived in such a way that the solution in the simplified domain closely approximates the solution on the original domain. The simplification of the geometry is advantageous from a computational point of view; for example, the generation of an adequate mesh becomes straightforward. The homogenized model is also able to handle developing flows and can capture boundary layers. By means of numerical experiments, we show that the developed model is accurate and suitable for use in real life applications. The speed-up is significant and in certain cases it allows us to handle problems that otherwise would be too computationally demanding.

In Chapter 6 we turn our attention to isothermal turbulent flow. Due to the very diverse difficulties that the problem of turbulence imposes, we take a pragmatic orientation in this case. We discuss the modeling of turbulence and present the (RANS) Reynolds Averaged Navier-Stokes equations. We consider two main models, the two-equation $k-\epsilon$ model and the algebraic type LVEL model. We show how the entropy production approach has to be modified for computing loss coefficients in turbulent flows with the $k-\epsilon$ model. A periodicity decomposition is used for reducing the high computational costs of this model. As an alternative, we present our implementation of the LVEL turbulence model for computing friction factors. This alternative proves to be more efficient and robust for handling turbulence. We provide comparison with experimental data on corrugated pipes and address the advantages and disadvantages of both methods. We conclude with Chapter 7, where we summarize the main results of this thesis and discuss some potential research topics.
Chapter 2

Governing Equations for Fluid Flow and Heat Transfer

In the present chapter we introduce the governing equations for fluid flow. We start with a short description of the conservation laws of mass, momentum and energy, which give place to the Navier-Stokes equations. Then we present a derivation of the Boussinesq approximation to the Navier-Stokes equations. Afterwards we scale the equations and present a dimensional analysis which in turn allows us to obtain some simplified models of the Navier-Stokes equations. We obtain the incompressible flow and the forced and natural convection approximations. Finally we discuss the role of friction factor and loss coefficients, and we show how to correctly determine these coefficients. The models presented in this chapter are the starting point for the techniques and methods which are derived in subsequent chapters for efficiently simulating flow in axisymmetric pipes. We consider it appropriate to suggest the following references for further reading on the topics presented in this chapter, we mention [64] for reviewing mathematical aspects, [55] for the physical perspective and [8] for a more applied point of view.

2.1 Fluid Mechanics

Fluid mechanics is a discipline which studies the flow of fluids and it is of interest in all kind of applications [15]. The mathematical foundations of fluid mechanics date back to 1738, when Daniel Bernoulli published his “Hydrodynamica” which may be considered the first fluid mechanics text. Further mathematical justification was provided
by Claude-Louis Navier (1822) and George-Gabriel Stokes (1822) in the Navier-Stokes equations which form the cornerstone of fluid mechanics [4].

2.2 Conservation Laws

The general equations describing the behavior of a fluid in motion are the well-known conservation laws of mass, momentum and energy [64]. First we introduce the convective derivative

$$\frac{D\psi}{Dt} := \partial_t \psi + u \cdot \nabla \psi,$$

where $\psi$ is a generic variable representing a scalar field. The material derivative can also be applied to a vector field, by interpreting $\nabla u$ as the covariant derivative [27]. The covariant derivative can be computed symbolically as $\nabla \otimes u$, were $\otimes$ is the tensor product (see Appendix A), however, for notation convenience we will write $\nabla u$ instead of $\nabla \otimes u$. The convective derivative symbol $D/Dt$ is used to emphasize that we are following a material volume while taking the derivative. Using the convective derivative, the conservation equations of mass, momentum and energy can be written in the following form

\begin{align}
\text{mass: } \frac{D\rho}{Dt} &= -\rho \nabla \cdot u, \quad (2.1a) \\
\text{momentum: } \rho \frac{Du}{Dt} &= \nabla \cdot T + \rho f, \quad (2.1b) \\
\text{energy: } \rho \frac{De}{Dt} &= T \cdot \nabla u - \nabla \cdot q + \Psi, \quad (2.1c)
\end{align}

where $u [m \, s^{-1}]$ is the flow velocity, $\rho [kg \, m^{-3}]$ the fluid density and $e [J \, kg^{-1}]$ the specific internal energy. The double-dot product between two tensors $A = (a_{ij})$ and $B = (b_{ij})$, is a scalar defined as $A : B := \Sigma_{ij} a_{ij} b_{ij}$ (see Appendix A). Equation (2.1b) it is known as Cauchy’s equation of motion [16] and it represents momentum conservation, where $T [Pa]$ is the stress tensor and $f [m \, s^{-2}]$ an external or volume force field acting on the material, in our particular case, we will consider the force exerted by gravity, i.e., $\rho g$, where $g [m \, s^{-2}]$ is the vector of gravitational acceleration. In (2.1c), $q [W \, m^{-2}]$ denotes the heat flux, and $\Psi [W \, m^{-3}]$ the heat source density. The heat source density can be used to model sources such as radioactive or electric heating [92].

2.2.1 Stress Decomposition

It is customary to split the stress tensor into a part depending on the thermodynamic pressure $p [Pa]$, representing the stationary normal components, and a viscous part $\tau$,
which depends on the fluid motion only. In fact, we will relate the viscous part to
the velocity gradients via the constitutive equations in a later section, meanwhile we
decompose
\[ \mathbf{T} = -p\mathbf{I} + \mathbf{\tau}, \]  
\[ \text{(2.2)} \]
where \( \mathbf{\tau} \) [Pa] is the viscous stress tensor. After this decomposition the system of equations
\((2.1)\) takes the following form

\[ \text{mass: } \frac{D\rho}{Dt} = -\rho \nabla \cdot \mathbf{u}, \]  
\[ \text{(2.3a)} \]
\[ \text{momentum: } \rho \frac{D\mathbf{u}}{Dt} = -\nabla p + \nabla \cdot \mathbf{\tau} + \rho g, \]  
\[ \text{(2.3b)} \]
\[ \text{energy: } \rho \frac{De}{Dt} = -p \nabla \cdot \mathbf{u} + \mathbf{\tau} : \nabla \mathbf{u} - \nabla \cdot \mathbf{q} + \Psi. \]  
\[ \text{(2.3c)} \]
The first, second and third term on the right hand side of \((2.3b)\) represent the change
in momentum due to the pressure gradient, viscous stresses and the action of gravity,
respectively. The right hand side terms of the energy equation \((2.3c)\) represent, the work
due to fluid expansion, the viscous dissipation, the addition of heat by conduction and
the addition of heat due to heat sources, in order from left to right. The system of
equations \((2.3)\) requires to be completed with models for the viscous stress tensor \( \mathbf{\tau} \) [Pa]
and for the heat flux \( \mathbf{q} \) [W m\(^{-2}\)]. In addition one requires two equations of state; one
relating the basic thermodynamic variables \( \rho, p \) and \( T \), and a second one relating the
specific internal energy \( e \) to the basic thermodynamic variables \( \rho, p \) and \( T \) (see Section
2.2.3).

2.2.2 Entropy Equation

Depending on the kind of application considered, it can be convenient to rewrite the
energy equation \((2.3c)\) in terms of the specific enthalpy \( h \) [J kg\(^{-1}\)], or the specific entropy
\( s \) [J kg\(^{-1}\) K\(^{-1}\)]. In our case we will proceed using the entropy \( s \). We will use this equation
later in order to rewrite the energy equation in terms of the absolute temperature \( T \) [K],
and in a posterior section, we will also use this equation for a discussion on pressure
losses (more precisely losses of total head) in pipe flow.

In order to rewrite the energy equation in terms of the entropy \( s \), we need to make use
of the first law of thermodynamics for a reversible process which in differential form
reads \[56\]
\[ Tds = de + p \, d\rho^{-1}. \]  
\[ \text{(2.4)} \]
Substituting this expression in the energy equation \((2.3c)\) and making use of \((2.3a)\) we
we can simplify the equation and obtain the equation in terms of the entropy $s$

\[ \rho T \frac{Ds}{Dt} = \tau : \nabla u - \nabla \cdot q + \Psi. \quad (2.5) \]

The entropy equation (2.5) will be very useful for the discussion on friction factors and loss coefficients in a later section.

### 2.2.3 Constitutive Equations

As we mentioned before, the system of equations describing conservation of mass (2.3a), conservation of momentum (2.3b) and conservation of energy (2.3c), is not complete and in order to provide a complete mathematical description, we need to add constitutive equations. More precisely, we need to provide models for the viscous stress tensor $\tau$ and for the heat flux vector $q$. In addition, two equations of state, one relating the basic thermodynamic variables $\rho, p$ and $T$, and another one relating the specific internal energy $e$ to the basic thermodynamic variables, need to be provided.

The constitutive equations for the viscous stress tensor $\tau$ of a *Newtonian fluid* and for the heat flux $q$ are

**Newton's viscous stress tensor**

\[ \tau = 2\mu \mathbf{D} + \lambda (\nabla \cdot \mathbf{u}) \mathbf{I}, \quad (2.6a) \]

**Fourier's heat flux**

\[ q = -\kappa \nabla T, \quad (2.6b) \]

where $\kappa$ [W m$^{-1}$ K$^{-1}$] is the coefficient of heat conduction, $\mu$ [Pa s] the dynamic viscosity and $\mathbf{D}$ the deformation velocity tensor defined as

\[ \mathbf{D} := \frac{1}{2} \nabla \mathbf{u} + \frac{1}{2} (\nabla \mathbf{u})^T. \quad (2.7) \]

The term $\lambda$ is the so-called second viscosity, but in the particular case of *incompressible flows* this term drops out as we will see below (see Appendix A). We would like to point out that in general the viscosity $\mu$ and the thermal conductivity $\kappa$, might depend on temperature, but in the context of this thesis, we will take these properties as constant.

In addition, one needs to introduce an equation of state, relating the thermodynamic variables, $T$, $p$ and $\rho$, in order to close the system of equations. The ideal gas law is a typical example, it reads

**ideal gas law**

\[ p = \rho RT, \quad (2.8) \]

where $R$ [J kg$^{-1}$ K$^{-1}$] is the specific gas constant. For an ideal gas, it is possible to write
the specific gas constant $\mathcal{R}$ as

$$\mathcal{R} = C_p - C_V,$$  \hfill (2.9)

where $C_V$ and $C_p$ are the specific heats at constant volume and at constant pressure, respectively. These quantities are defined as

$$C_V := T \left( \frac{\partial s}{\partial T} \right)_\rho$$

and

$$C_p := T \left( \frac{\partial s}{\partial T} \right)_p.$$  \hfill (2.10)

The subscript $\rho$ means that the partial derivative is taken at constant density. In the same way, the subscript $p$ means that the partial derivative is taken at constant pressure. Under the assumption of an ideal gas, the specific internal energy $e$ depends only on $T$, more precisely we have [130]

$$e = C_V T.$$  \hfill (2.11)

Typically, the set of equations (2.3) completed with the constitutive equations (2.6), with an equation of state (typically the ideal gas law (2.8)) and with (2.11), is referred to as the Navier-Stokes equations. However, there is no uniform nomenclature, and sometimes this name is also used to denote the momentum equation (2.3b) only [64]. The ideal gas law (2.8) is a good approximation to the behavior of many gases under many conditions, although it has several limitations. In our particular case, we will not appeal directly to the ideal gas law, instead, we will use a Taylor expansion of the density as function of $p$ and $T$. We explain this in more detail in Section 2.3.1.

### 2.3 Oberbeck-Boussinesq Approximation

In the present section we will address the Boussinesq approximation to the Navier-Stokes equations. The main goal of the Boussinesq approximation is to be able to describe the effects of buoyancy with a convenient set of quasi-incompressible equations. In fact, the buoyancy forces appear due to the variations of density within the fluid, when the fluid is under the influence of the gravitational field. The appearance of these forces becomes clear when an equation of state is inserted into the equations of motion (2.3b). However, even though the density variations give rise to buoyant forces, it is possible to address the fluid as incompressible in the sense that the variations of density can be ignored except were they are multiplied by the acceleration of gravity in the momentum equation. This was written by Boussinesq himself in [11] and this system of equations is mostly know after his name, however this system of equations seem to have been first used by Oberbeck [79].

The Boussinesq approximation has been know for more than 100 years, however, formal
asymptotic derivations of the Boussinesq approximations are still a current subject for publication [94, 131]. In our case, we will derive these equations in an intuitive manner. The derivation presented is based on Kundu [55] and Landau [56]. For a more comprehensive review on asymptotic models for fluid flow we refer the reader to Zeytounian [132] and to Gray [34].

### 2.3.1 Thermal Expansion and Compressibility

The basic assumption on which the Boussinesq approximation is based, is the quasi-incompressibility of the fluid. Generally speaking, this means that the pressure is supposed to vary only slightly through the fluid, so that density changes due to changes in pressure may be neglected. This already implies a restriction on the maximum height (measured in the direction of gravity), because a very large height would imply a large change in hydrostatic pressure and hence the approximation would not be valid anymore [56]. In mathematical terms, the condition which we require is

$$\frac{1}{\rho} \left( \frac{\partial \rho}{\partial p} \right)_T (p - p_{\text{ref}}) \ll 1, \quad (2.12)$$

where $p_{\text{ref}}$ [Pa] is a reference pressure. The density changes due to temperature changes, cannot in general be neglected. For example, in problems involving natural convection, the density differences are responsible for driving the fluid. Assuming that the temperature differences in the fluid are not too large, and that (2.12) holds, we can approximate the density by the following expression

$$\rho = \rho_{\text{ref}} \left( 1 - \alpha_V (T - T_{\text{ref}}) \right), \quad (2.13)$$

where $T_{\text{ref}}$[K] is a reference temperature, $\rho_{\text{ref}}$[kg m$^{-3}$] is the density of the fluid at the reference temperature and $\alpha_V$[K$^{-1}$] is the **volumetric thermal expansion coefficient** defined as (see [130])

$$\alpha_V := -\frac{1}{\rho} \left( \frac{\partial \rho}{\partial T} \right)_p. \quad (2.14)$$

The volumetric thermal expansion coefficient $\alpha_V$ is also evaluated at the reference temperature $T_{\text{ref}}$. Typically, this reference temperature is taken from the boundary conditions of the problem. This just means that the expansion point $T_{\text{ref}}$ of the linearization (2.13) really represents the temperatures characteristic for the problem. The partial derivative is taken while keeping the pressure constant, i.e., $p = p_{\text{ref}}$. This means that (2.13), is in fact a 1st order Taylor expansion of the density at the point $T = T_{\text{ref}}$, $p = p_{\text{ref}}$, where the changes in density due to pressure have been neglected on basis of (2.12). In Section 4.4.2, we shown how this simple Taylor expansion is very accurate for approximating the density of liquefied methane, which has a coefficient of volumetric expansion $\alpha_V = 3.7 \times 10^{-3}$[K$^{-1}$]. The coefficient of volumetric expansion $\alpha_V$ is typically
2.3 Oberbeck-Boussinesq Approximation

very small and we will use this fact in order to obtain the Boussinesq approximation.

2.3.2 Continuity Equation

Now we will use the quasi-incompressibility assumption to justify why it is possible to neglect the term $\rho^{-1}(D\rho/ Dt)$ in equation (2.3a). Due to (2.13) we know that the typical change in density $\Delta\rho$ satisfies

$$\frac{|\Delta\rho|}{\rho_{\text{ref}}} = \alpha_V |\Delta T| \ll 1,$$

(2.15)

with $\alpha_V$ being typically very small. For example, methane at a temperature of $106.95\,[\text{K}]$ and under a pressure of $9 \times 10^5\,[\text{Pa}]$ has a volumetric expansion coefficient of $\alpha_V = 3.7 \times 10^{-3}\,[\text{K}^{-1}]$ [58, 104]. This means that even for a change in temperature of $10\,[\text{K}]$, the density would change by less than 4 percent. Thus, provided the temperature difference $\Delta T$ is not too large, we can conclude $|\Delta\rho|/\rho_{\text{ref}} \ll 1$ as it was stated in (2.15).

Now we proceed to scale equation (2.3a). For this purpose let us assume the following typical scales for density, velocity, time and length

$$\rho \text{ with } \rho_{\text{ref}}, \Delta\rho; \quad u \text{ with } u_0; \quad x \text{ with } l_0; \quad t \text{ with } t_0 := \frac{l_0}{u_0}. \quad (2.16)$$

Notice that for the density we consider two scales, $\rho_{\text{ref}}$, shows the order of magnitude of the primary density and $\Delta\rho$ denotes a typical difference in density, which in the end are responsible for driving natural convecting flows. Under these assumptions we can rewrite (2.3a) in the following dimensionless form

$$\frac{\Delta\rho}{t_0} \frac{\partial}{\partial t} \rho^* + \frac{u_0}{l_0} \frac{\partial \rho^*}{\partial x} = -\frac{u_0}{l_0} (\rho_{\text{ref}} + \Delta \rho) \nabla^* \cdot u^*, \quad (2.17)$$

where $u^* := u/u_0, t^* := t u_0/l_0$ and $\rho^* := (\rho - \rho_{\text{ref}})/\Delta\rho$ are the dimensionless velocity, time and density, respectively. The symbol $\nabla^*$ refers to the dimensionless gradient operator. Multiplying by $l_0/(u_0 \rho_{\text{ref}})$ and using that $t_0 = l_0/u_0$, we obtain

$$\frac{\Delta\rho}{\rho_{\text{ref}}} \left( \partial_t \frac{\partial}{\partial t} \rho^* + u^* \cdot \nabla^* \rho^* + \rho^* \nabla^* \cdot u^* \right) = -\nabla^* \cdot u^*. \quad (2.18)$$

Since the variables are dimensionless and $|\Delta\rho|/\rho_{\text{ref}} \ll 1$ according to (2.15) we can neglect the left hand side term in (2.18) and we can replace the continuity equation by the incompressible form, which written back in dimensional form reads

$$\text{Boussinesq continuity: } \nabla \cdot u = 0. \quad (2.19)$$
2.3.3 Momentum Equations

The incompressibility condition (2.19) which we derived in the previous section allows us to eliminate the second viscosity term $\lambda(\nabla \cdot u)\mathcal{I}$ from the viscous stress tensor (see 2.6) yielding

$$\tau = 2\mu D. \tag{2.20}$$

With this condition the momentum equations (2.3b) take the form (see Appendix, A eq.(A.9) for more details)

$$\rho \frac{Du}{Dt} = -\nabla p + \mu \nabla^2 u + \rho g. \tag{2.21}$$

Now we will consider a hypothetical static reference state in which the density is $\rho_{ref}$ everywhere. This reference state has an associated hydrostatic pressure field $p_h$ satisfying $\nabla p_h = \rho_{ref}g$, i.e., the hydrostatic field of a fluid with constant density $\rho_{ref}$. For example, if the gravity vector $g$ is oriented in the direction $-e_x$, then $p_h = p_h(x)$. Now we write $p = p_h + p'$ and $\rho = \rho_{ref} + \rho'$, where $p'$ and $\rho'$ represent the deviation from the hydrostatic pressure and the deviation from the reference density, respectively. Substituting these expression in (2.21) and using that $p_h = p_h(x)$, we obtain

$$\left(\rho_{ref} + \rho'\right) \frac{Du}{Dt} = -\nabla p' + \mu \nabla^2 u + \rho' g. \tag{2.22}$$

Dividing the previous equation by $\rho_{ref}$ we get

$$\left(1 + \frac{\rho'}{\rho_{ref}}\right) \frac{Du}{Dt} = -\frac{1}{\rho_{ref}} \nabla p' + \nu \nabla^2 u + \frac{\rho'}{\rho_{ref}} g, \tag{2.23}$$

where $\nu := \mu/\rho_{ref}[m^2s^{-1}]$ is the kinematic viscosity of the fluid. The quotient $\rho'/\rho_{ref}$ appears in the inertia term and in the buoyancy term. Assuming again that the variations in temperature are not too large, the density variations in the inertia term can be neglected. However, the term $\rho' g/\rho_{ref}$ can not always be neglected, this will depend on the scaling for the velocity $u$. For example, in natural convection problems this term is vital because it is the one that drives the flow. We will discuss this in more detail in a later section. The simplified momentum equations which we just obtained after substituting (2.13) become

Boussinesq momentum: $\rho_{ref} \frac{Du}{Dt} = -\nabla p + \mu \nabla^2 u - \rho_{ref} \alpha_V (T - T_{ref}) g. \tag{2.24}$

where from now on, $p$ denotes the deviation from the hydrostatic pressure $p_h$. 

Now we will proceed to rewrite the energy equation (2.3c) in terms of the temperature \( T \). Discarding heat sources, the energy equation reads

\[
\frac{\partial e}{\partial t} = \kappa \nabla^2 T - p \nabla \cdot u + \tau : \nabla u. \tag{2.25}
\]

Now we will make use again of the quasi-incompressibility assumption, i.e., we will use the fact that the density is not very sensitive to changes in pressure. Under this assumption we have \( d\rho \approx \left( \frac{\partial \rho}{\partial T} \right)_p dT \), which in turn allows us to write

\[
-p \nabla \cdot u = \frac{p \rho}{\rho} \frac{\partial \rho}{\partial T} \frac{dT}{dt} \approx -p \alpha_V \frac{dT}{dt}. \tag{2.26}
\]

Assuming the ideal gas relation (see (2.8)), for which \( R = C_p - C_V \), \( \alpha_V = 1/T \), the previous relation becomes

\[
-p \nabla \cdot u = -\rho R T \alpha_V \frac{dT}{dt} = -\rho (C_p - C_V) \frac{dT}{dt}. \tag{2.27}
\]

Finally, substituting the previous expression in (2.25) and using the fact that for an ideal gas \( e = C_V T \) (see (2.11)), the energy equation in the Boussinesq approximation takes the form

\[
\text{Boussinesq temperature: } \rho C_p \frac{dT}{dt} = \kappa \nabla^2 T + \tau : \nabla u, \tag{2.28}
\]

where \( \tau = 2\mu \mathcal{D} \). In this equation, it is customary to replace \( \rho \) on the left hand side by \( \rho_{ref} \). This substitution is adequate for natural convection problems in which the temperature differences are not too large [8].

In summary, the Boussinesq approximation allows us to handle the density as constant in the continuity and in the momentum equations excluding the gravity term. The properties of the fluid \( \mu, \kappa \) and \( C_p \) are assumed constant. One should keep in mind that the system (2.19), (2.24) and (2.28) is an approximation and as such it can not be used in all cases. Generally speaking, the approximation is appropriate if the Mach number of the flow is small, propagation of sound or shock waves are not considered, the vertical scale of the flow is not too large and the temperature differences in the fluid are not too large [55].
2.4 Dimensional Analysis

In this section we will present a dimensional analysis for the equations of the Boussinesq approximation. Based on this analysis we will obtain the equations for isothermal flow, forced convection and natural convection. Before doing this we summarize the equations of the Boussinesq approximation

continuity: \( \nabla \cdot \mathbf{u} = 0 \), \hfill (2.29a)

momentum: \( \rho_{\text{ref}} \frac{D\mathbf{u}}{Dt} = -\nabla p + \mu \nabla^2 \mathbf{u} - \rho_{\text{ref}} \alpha_v (T - T_{\text{ref}}) \mathbf{g} \), \hfill (2.29b)

temperature: \( \rho_{\text{ref}} C_p \frac{DT}{Dt} = \kappa \nabla^2 T + \tau : \nabla \mathbf{u} \), \hfill (2.29c)

where \( \rho_{\text{ref}} \) is the reference density corresponding to the reference temperature \( T_{\text{ref}} \), and reference pressure \( p_{\text{ref}} \). These can be taken from the boundary conditions of the problem, for example.

Now we assume a typical length scale \( l_0 \) and velocity \( u_0 \). Using these scales we define the following dimensionless variables

\[ x^* := \frac{x}{l_0}, \quad t^* := \frac{u_0}{l_0} t, \quad u^* := \frac{u}{u_0}, \quad p^* := \frac{p - p_{\text{ref}}}{\rho_{\text{ref}} u_0^2}, \quad T^* := \frac{T - T_{\text{ref}}}{\Delta T}, \quad \tau^* := \frac{l_0}{u_0 \mu} \tau, \] \hfill (2.30)

where \( \Delta T \) represents a typical temperature difference in the problem (which can be determined from the boundary conditions for instance). The definitions of the dimensionless variables induce the following differentiation rules

\[ \nabla^* = l_0 \nabla, \quad \frac{D}{Dt^*} = \frac{l_0}{u_0} \frac{D}{Dt} \] \hfill (2.31)

In terms of the dimensionless variables defined above, the Boussinesq system (2.29) takes the form

continuity: \( \nabla^* \cdot \mathbf{u}^* = 0 \), \hfill (2.32a)

momentum: \( \frac{D\mathbf{u}^*}{Dt^*} = -\nabla^* p^* + \left( \frac{\mu}{l_0 u_0 \rho_{\text{ref}}} \right) \nabla^* \mathbf{u}^* - \left( \frac{g l_0 \alpha_v \Delta T}{u_0^2} \right) \mathbf{g} T^*, \) \hfill (2.32b)

temperature: \( \frac{DT^*}{Dt^*} = \left( \frac{\kappa}{l_0 u_0 \rho_{\text{ref}} C_p} \right) \nabla^* \mathbf{u}^* \) \hfill (2.32c)

where \( g = 9.806 \text{[m/s}^2\text{]} \) is the gravitational acceleration (\( g = |g| \)). The characteristic velocity can be chosen in several ways depending on the type of problem and this will allow us to distinguish between forced and natural convection. The dimensionless groups ap-
pearing in (2.32) can be interpreted as ratios of various forces or effects in the system. In order to see this clearly, it is convenient to introduce the following dimensionless numbers

\[
Re := \frac{l_0 u_0 \rho_{ref}}{\mu} = \frac{l_0 u_0}{\nu}, \quad Gr := \frac{g \alpha \nu^2}{\nu^2}, \quad Pr := \frac{C_p \mu}{\nu}, \quad Br := \frac{\mu u_0^2}{\nu \kappa}.
\]

where \(Re\), \(Gr\), \(Pr\) and \(Br\) numbers, are the Reynolds, Grashof, Prandtl and Brinkman numbers, respectively. We also mention that the Prandtl number \(Pr\) is a property of the fluid and not of the flow. Making use of these dimensionless numbers, (2.32) can be written as

\[
\begin{align*}
\text{continuity:} \quad & \nabla \cdot \mathbf{u}^* = 0, \quad \text{(2.34a)} \\
\text{momentum:} \quad & \frac{D\mathbf{u}^*}{Dt^*} = -\nabla \cdot p^* + \frac{1}{Re} \left(\nabla \cdot \mathbf{u}^* - \frac{Gr}{Re} \frac{g}{\nu^2} T^*ight), \quad \text{(2.34b)} \\
\text{temperature:} \quad & \frac{DT^*}{Dt^*} = \frac{1}{Re Pr} (\nabla \cdot \mathbf{u}^*) T^* + \frac{Br}{Re Pr} \tau^* : \nabla \mathbf{u}^*. \quad \text{(2.34c)}
\end{align*}
\]

The dimensionless numbers now give us an idea of the balance between various terms in the system. The system of equations (2.34) is very useful for heat transfer analysis because it allows us to distinguish between the limiting cases of forced convection and natural (free) convection. The difference between these two cases comes from the choice that is made for the velocity scale \(u_0\). In the case of forced convection, the buoyancy term \(\rho_{ref} g \alpha \nu (T - T_{ref})\) in the momentum equations (2.29b) can be neglected. In the case of natural convection, this term cannot be neglected. The viscous dissipation term \(\tau : \nabla \mathbf{u}\) in the temperature equation (2.29c) can be neglected for the application which we consider in this thesis, however the argumentation in the case of forced and natural convection are slightly different. When both, forced and natural convection take place, the problem is referred to as a mixed convection problem [6].

Other dimensionless numbers which are very common for describing non-isothermal systems are the Péclet number \(Pe\), the Rayleigh number \(Ra\) and the Eckert number \(Ec\) defined as

\[
Pe := Re Pr, \quad Ra := Gr Pr, \quad Ec := \frac{Br}{Pr}.
\]

In order to gain some insight in the meaning of the dimensionless numbers (2.33), it is convenient to interpret these numbers as ratios of various forces or effects in the system, as in Table 2.1.
2.4.1 Forced Convection

We will now obtain the equations in the limiting case of forced convection. The term forced convection refers to a type of heat transport in which the fluid motion is generated by an external source, a pump for instance. In other words, the velocity scale \( u_0 \) is imposed by a boundary condition. Under the term forced convection one typically means that \( u_0^2 \gg g_0 \alpha_Y \Delta T \). In physical terms, this assumption means that the buoyancy speed \((g_0 \alpha_Y \Delta T)^{1/2}\) is negligible compared to the imposed (forced) speed \( u_0 \). In terms of the dimensionless numbers, this assumption reads \( \text{Gr/Re}^2 \ll 1 \).

The viscous dissipation term can in general be neglected from the temperature equation (2.34c), the condition is that the velocity \( u_0 \) should not be too large. To be more specific we consider the case of methane at a temperature of 100[K] and at a pressure of 9 × 10^5[Pa]. Under these conditions, we have \( \mu = 1.57 \times 10^{-4}[\text{Pa.s}] \), \( \kappa = 0.2[\text{W m}^{-1} \text{K}^{-1}] \), and \( \rho_{\text{ref}} = 439[\text{kg m}^{-3}] \). For this particular case we can write the condition \( Br \ll 1 \) explicitly as

\[
\frac{u_0^2}{\Delta T} \ll 1282[\text{m}^2 \text{s}^{-2} \text{K}^{-1}].
\]

For example, when the characteristic temperature difference is \( \Delta T \approx 0.1[\text{K}] \), in order to neglect the viscous dissipation term in (2.34c), we require \( u_0 \ll 11[\text{m s}^{-1}] \), which in terms of the Reynolds number (for a diameter of 0.02[m]) translates into the condition \( \text{Re} \ll 5.8 \times 10^5 \), which is very reasonable for many applications.
Altogether, the system of equations describing forced convection reduces to

\begin{align}
\text{continuity:} \quad \nabla \cdot \mathbf{u}^* &= 0, \quad (2.37a) \\
\text{momentum:} \quad \frac{D\mathbf{u}^*}{Dt^*} &= -\nabla^* p^* + \frac{1}{Re} \nabla^* 2 \mathbf{u}^*, \quad (2.37b) \\
\text{temperature:} \quad \frac{DT^*}{Dt^*} &= \frac{1}{Pe} \nabla^* 2 T^*. \quad (2.37c)
\end{align}

We notice that in (2.37) the energy equation is decoupled from the continuity and the momentum equations. In fact, the system formed by equations (2.37a) and (2.37b) only, is referred to as the incompressible Navier-Stokes equations. In our case, we have obtained this system as a particular case of the Boussinesq approximation, however it is also possible to derive it, if one assumes a constant density [56, pp.17].

### 2.4.2 Natural Convection

In the case of natural convection, the velocity is not imposed by an external source, instead, it arises from the temperature differences within the fluid. Since there is no imposed velocity, we need to scale the velocity with other parameters. In some references, for instance in [8] and [117], the velocity is scaled as \( u_0 = \nu/l_0 \) for natural convection problems. However, as pointed out by Perez [84], Gray [34] and Principe [94], we also believe that it is more appropriate to scale the velocity as

\[
\nu_0 = (\alpha V \Delta T gl_0)^{1/2}. \quad (2.38)
\]

The previous scaling of the velocity represents a buoyancy speed which (in contrast to the viscous speed \( \nu/l_0 \)) shows the correct behavior with respect to \( \Delta T \), i.e., the velocity increases with \( \Delta T \). Once we have chosen (2.38) as our characteristic velocity, the Brinkman number takes the form

\[
\text{Br} = \frac{\mu \nu_0^2}{\kappa \Delta T} = \frac{\mu \alpha V g l_0}{\kappa} \ll 1. \quad (2.39)
\]

The Brinkman number in this kind of application depends only on properties of the fluid and on a the typical length scale \( l_0 \). The combination \( \mu \alpha V g/\kappa \) is typically very small. For example, for methane at a temperature of 100[K] and at a pressure of \( 9 \times 10^5[Pa] \), we have \( \mu = 1.57 \times 10^{-4}[Pa s], \alpha_V = 3.7 \times 10^{-3}[K^{-1}] \) and \( \kappa = 0.2[W m^{-1}K^{-1}] \), which yields a Brinkman number of \( \text{Br} \approx 10^{-5}[m^{-1}]l_0 \). This means that the viscous heating term \( \tau^* \text{ : } \nabla^* \mathbf{u}^* \) can be neglected from (2.29c), provided that the length scale of the system is not very large, \( l_0 \lesssim 10[m] \) would be a sufficient condition for example.
On the other hand, (2.38) implies that
\[
\frac{Gr}{Re^2} = \frac{g\alpha_0 V \Delta T}{u_0} = 1.
\] (2.40)

Altogether the system of equations describing natural convection becomes

continuity: \( \nabla^* u^* = 0 \), \hfill (2.41a)

momentum: \( \frac{Du^*}{Dt^*} = -\nabla^* p^* + \frac{1}{Re} \nabla^* u^* \Delta T^* - \frac{g}{g} \), \hfill (2.41b)

temperature: \( \frac{DT^*}{Dt^*} = \frac{1}{Pe} \nabla^* T^* \), \hfill (2.41c)

where the Reynolds number satisfies \( Re^2 = Gr \), according to (2.40). This system of equations can also be written in terms of the Rayleigh number \( Ra \), by using the relation \( Ra = Gr Pr \). If this is done, then the term \( 1/Re \) in (2.41b) and the term \( 1/Re Pr \) in (2.41c) become \( (Pr/Ra)^{1/2} \) and \( 1/(Pr Ra)^{1/2} \), respectively [34].

### 2.4.3 Boundary Conditions for Fluid Flow

In the previous sections, we have presented the governing equations for fluid flow and heat transfer. These equations find their basis in the laws of conservation of mass, momentum and energy (2.1). We also introduced the Boussinesq approximation (2.29), which allows us to describe natural convecting flows, and we also presented some simplifications. The corresponding system of equations should hold at any point inside the domain of interest \( \Omega \). However, we have not yet said anything about how the fluid flowing through the domain \( \Omega \) interacts with the surroundings, nor anything about the initial state of the system.

In fact, the system of equations (2.29) is a system of evolution equations. Therefore, if we want to resolve an initial value problem, it is necessary to prescribe a set of initial conditions for \( u(x, t), p(x, t), \) and \( T(x, t) \), at the initial time \( t = 0 \). The interactions with the surroundings are described using suitable boundary conditions at the boundary of the domain \( \partial \Omega \). For the fluid dynamic part of the problem, either the velocity components or the total surface stress must be specified on the boundary of the fluid region [97]. The boundary conditions are typically classified as

*Dirichlet* or essential boundary conditions

\[
u = f_u \quad \text{on} \quad \Gamma_u, \quad (2.42)
\]
*Neumann* or natural boundary conditions

\[ \mathbf{T} \mathbf{n} = f_T \quad \text{on} \quad \Gamma_T, \quad (2.43) \]

where \( f_u \) and \( f_T \) are vector functions defined on \( \Gamma_u \) and \( \Gamma_T \), respectively. The sets \( \Gamma_u \) and \( \Gamma_T \) denote the parts of the boundary on which a Dirichlet and Neumann boundary condition are prescribed, respectively. The boundary of the whole domain \( \partial \Omega \) can be written as \( \partial \Omega = \Gamma_u \cup \Gamma_T \). In the context of an initial value problem \( f_u \) and \( f_T \), might be functions of time as well.

As a particular case of the Dirichlet boundary condition (2.42), we have the homogeneous Dirichlet velocity boundary condition,

\[ \text{No-slip:} \quad u = 0, \quad \text{on} \quad \Gamma, \quad (2.44) \]

where \( \Gamma \subset \Gamma_u \) is a section of the boundary with Dirichlet conditions. The homogeneous Dirichlet boundary condition for the velocity field is more commonly referred to as the *no-slip boundary condition*, expressing that the fluid adheres to the physical boundary \( \Gamma \). Thus \( \Gamma \) represents the contact region between the fluid and a solid impermeable surface, the wall of a pipe for example. A non-homogeneous Dirichlet boundary condition is typically used to prescribe an inflow velocity profile.

The Neumann boundary condition (2.43) can be used to simulate an outflow boundary, which corresponds to a zero force at the outlet. This Neumann boundary condition acts as an approximate non-affecting outflow boundary condition and it is also referred to as a do-nothing boundary condition or *natural boundary condition* \[44,96\], this condition reads

\[ \mathbf{T} \mathbf{n} = 0. \quad (2.45) \]

There are also some alternatives to the previous boundary conditions. One leads to the possibility of representing periodic boundary conditions in a flow. For example, for a geometry which is periodic in the direction \( e_x \), with period \( L \), the *periodic boundary conditions* for the velocity and the stress read

\[ u(x,t) = u(x + L e_x, t), \quad (2.46a) \]

\[ \mathbf{T}(x,t)\mathbf{n}(x) = \mathbf{T}(x + L e_x, t)\mathbf{n}(x + L e_x) + f_T(x + L e_x, t). \quad (2.46b) \]

The vector \( L e_x \) indicates the spatial offset from one boundary to another. \( L \) is the period, and \( e_x \) represents the direction in which the flow is periodic. The velocity components on the periodic surfaces are equal but unknown. The stress on the two surfaces may differ by a function \( f_T \). The most common situation is for the normal stresses (essentially the pressure) to have an offset equal to the pressure change over the length \( L \).
while the tangential stresses are equal [97].

For inviscid fluids one can impose a slip boundary condition, which corresponds to setting the normal component of the velocity to zero, i.e., \( u \cdot n = 0 \). The slip boundary condition also requires the stress vector to be normal to the surface, i.e., \( t \cdot T n = 0 \), for any tangential vector \( t \). The slip boundary condition models an impermeable boundary with negligible friction [44].

### 2.5 Flow in Pipes

Now we turn our attention to the main subject of this thesis, namely flow in pipes and conduits. Liquid or gas flow through pipes is commonly used in distribution networks, and for heating and cooling applications. The fluid can be forced to flow by an external device (a fan or a pump for instance) or the driving mechanism can be natural convection [6]. In the case of forced flow (see Section 2.4.1), a very important quantity in the analysis of pipe flow is the so-called pressure drop. In the case of fully developed flow in a straight pipe, the pressure drop is directly related to the mean flow rate and it determines the power requirements of the device to maintain the flow.

#### 2.5.1 The Darcy Friction Factor

For isothermal fully developed flow in a straight pipe, the pressure drop \( \Delta p \) is usually expressed in terms of the friction factor. The friction factor is a non-dimensional representation of the pressure drop which (under this regime) is exactly balanced by skin friction forces [41]. The skin friction appears due to the no-slip condition at the surface of the pipe. Because of this condition, the velocity in a pipe changes from zero at the surface to a maximum at the pipe center. In fluid flow, it is convenient to work with the cross-sectional average velocity \( U_x \), which remains constant for incompressible flows when the cross-sectional area of the pipe is constant, as in the case of a straight pipe for example.

In practice, the friction factor is used to describe the pressure loss for all types of fully developed internal flows (laminar or turbulent flows, smooth or rough surfaces) via the relation [15]

\[
\Delta p = f \frac{L \rho U_x^2}{D},
\]

(2.47)

where, \( \Delta p = p_{in} - p_{out} \) is the pressure drop over a segment of length \( L \), \( f \) is the Darcy friction factor, \( D \) is the diameter of the pipe, \( \rho \) is the density and \( U_x \) is the average velocity over the cross section of the pipe, as depicted in Figure 2.1. In the case of laminar
2.5 Flow in Pipes

Figure 2.1: Poiseuille flow in a straight pipe with length $L$, diameter $D$ and average velocity $U_x$. The pressure drop is given by $\Delta p = p_{\text{in}} - p_{\text{out}}$.

Poiseuille flow (i.e., for Poiseuille flow, see Figure 2.1), it is possible to compute the friction factor analytically and we obtain

$$ f = \frac{64}{\text{Re}} $$

(2.48)

where $\text{Re}$ is the Reynolds number defined in (2.33), with $u_0 = U_x$ and $l_0 = D$. This simply means that we have chosen the cross-sectional average velocity and the pipe diameter as the typical scales for velocity and length respectively, i.e.,

$$ \text{Re} := \frac{u_0 l_0}{\nu} = \frac{U_x D}{\nu}. $$

(2.49)

Even though the losses in internal flows are commonly called pressure losses, what they truly represent are losses of total head because they occur when the total pressure, i.e., the mechanical energy in the flow, is reduced [41]. In other words, the mechanical energy of the flow is dissipated into internal energy. In fact, equation (2.47) constitutes a lumped model for steady flow in a horizontal pipe with constant cross-section. Having a horizontal pipe warranties that there is no change in potential energy and having a constant cross-section ensures having a constant average axial velocity $U_x$. Under this setting, $\Delta p/\rho$ represents the specific dissipation, i.e., the power lost due to dissipation. The more general cases will be discussed in Section 2.6.1, but first we will address the problem of periodic geometries.

2.5.2 Corrugated Pipes

When the radius of the pipe is not constant, for instance due to wall-roughness, the situation becomes more complicated. For instance, in the case of turbulent flow, the friction factor is found to be strongly dependent on the wall-roughness. Since wall-roughness is
to a high extend random, there is no way of characterizing all the possible wall details that can arise in practice, and therefore an equivalent sand roughness was introduced in order to be able to estimate the pressure drop in rough pipes. The typical way of doing flow calculations in this case is to make use of the Moody diagram [72] which is based on experiments performed on artificially roughened pipes [78]. The typical Moody diagram shows the friction factor as independent of roughness for laminar flow, however this is not correct and there are several numerical and experimental studies which have shown that the contribution of wall-shape is not trivial, even in the laminar case. This happens specially when dealing with a specific wall-shape design (as in the case of corrugated pipes), rather than just with some “small” random roughness [9,39,47,59].

Corrugated pipes are pipes in which the wall is designed to vary along the axial direction, these variations along the wall are referred to as corrugations. Due to their wall design, corrugated pipes respond very well to bending and are to some extend flexible. The flexibility allows for easy installation and also makes this kind of pipes suitable for dynamic applications, such as in offshore transfer of LNG [26].

For describing these kind of components, we consider an arbitrary axially symmetric geometry depicted as in Figure 2.2. The wall of the pipe $\Gamma$ can be described in terms of the cylindrical basis vectors $e_\phi$(radial), $e_\phi$(azimuthal) and $e_x$(axial) via the parametrization $x(\phi,x) = R_w(x)e_r + xe_x$, with $0 \leq \phi < 2\pi$, $0 \leq x \leq L_p$. We assume the wall radius $R_w(x)$ to be a smooth function of the axial coordinate $x$, consequently the outer unit normal vector $n$, and the surface element $dS$ can be expressed as

$$n = \frac{e_r - R'_w(x)e_x}{\sqrt{1 + R'_w(x)^2}}$$

$$dS = R_w(x)\sqrt{1 + R'_w(x)^2}d\phi dx,$$
where $(\cdot)'$ stands for the derivative with respect to the axial variable $x$. We note that the previous description is quite general and it allows us to describe not only periodic shapes, which are typical for corrugated pipes (see Figure 2.3), but also non-periodic axially symmetric geometries, such as gradual expansions and contractions.

The boundary condition for the velocity at the pipe’s wall which we consider is the no-slip boundary condition (2.44), which we recall here for clarity

$$u = 0 \quad \text{at} \quad \Gamma. \quad (2.51)$$

### 2.5.3 Integral Expression for the Friction Factor

It is still possible to use (2.47) for describing the flow in a horizontal pipe with varying radius. In order to do this, we have to select a characteristic diameter and an average velocity in the problem. One option is to take the values at the inlet of the pipe, i.e., $D = 2R_w(0)$ and

$$U_x = \frac{1}{\pi R_w^2(0)} \int_{\Gamma_{in}} u_x dS. \quad (2.52)$$

In the particular case of a periodic pipe, we set the geometry in such a way that the minimum radius is located at $x = 0$ (see Figure 2.3). Once we have selected our reference diameter and velocity, we can use (2.47) as a lumped model for describing the flow in any kind of pipe. The main difficulty is to efficiently determine the correct friction factor, i.e., the one that accurately predicts the pressure drop.

Now we will obtain an integral expression for the friction factor. For doing this, we need to make use of the incompressible Navier-Stokes equations (2.37a) and (2.37b), which in cylindrical coordinates and in steady state conditions ($\partial/\partial t = 0$) read

continuity: $\partial_x u_x + \partial_r u_r + \frac{1}{r} u_r = 0. \quad (2.53a)$

axial momentum: $u_x \partial_x u_x + u_r \partial_r u_x = \nu \left( \partial_{xx} u_x + \partial_{rr} u_x + \frac{1}{r} \partial_r u_x \right) - \frac{1}{\rho} \partial_x p. \quad (2.53b)$

radial momentum: $u_x \partial_x u_r + u_r \partial_r u_r = \nu \left( \partial_{xx} u_r + \partial_{rr} u_r + \frac{1}{r} \partial_r u_r \right) - \frac{1}{\rho} \partial_r p. \quad (2.53c)$

where $x$ denotes the axial coordinate, $r$ the radial coordinate, $u_x = u_x(r, x)$ the axial velocity, $u_r = u_r(r, x)$ the radial velocity, i.e., $\mathbf{u} = u_x e_x + u_r e_r$, and $p = p(r, x)$ the pressure. The azimuthal variable $\phi$, and velocity component $u_\phi$ do not play a role under the assumption of axially symmetric flow.

If we integrate the axial momentum equation (2.53b) over the domain $\Omega$, we can obtain
an expression for the pressure loss in terms of surface integrals over the pipe wall $\Gamma$. To this purpose, we first rewrite (2.53b), in the following form

$$\nabla \cdot \left( u \left( u \cdot u \right) \right) = -\frac{1}{\rho} \nabla \cdot \left( pe \right) + \nu \nabla \cdot \left( \nabla \cdot u \right),$$  \hspace{1cm} (2.54)$$

where we used $\partial_x p = \nabla \cdot \left( pe \right)$, and $u \cdot \nabla u = \nabla \cdot \left( u \cdot u \right)$. Integrating over the domain $\Omega$, see Figure 2.2, and applying the divergence theorem we get

$$\oint_{\partial \Omega} u \cdot u \cdot n \cdot dS = -\frac{1}{\rho} \oint_{\partial \Omega} p n \cdot dS + \nu \oint_{\partial \Omega} \frac{\partial u_x}{\partial n} dS,$$  \hspace{1cm} (2.55)$$

where $n_x = n \cdot e_x$. Next, we split the surface of integration $\partial \Omega = \Gamma_{in} \cup \Gamma_{out} \cup \Gamma$, as sketched in Figure 2.2. After using the no-slip condition (2.51), and rearranging terms we get

$$\int_{\Gamma_{in}} p \cdot dS - \int_{\Gamma_{out}} p \cdot dS = \rho \left[ \int_{\Gamma_{out}} u_x^2 \cdot dS - \int_{\Gamma_{in}} u_x^2 \cdot dS \right] +$$

$$+ \int_{\Gamma} p n_x \cdot dS - \mu \oint_{\partial \Omega} \frac{\partial u_x}{\partial n} dS.$$  \hspace{1cm} (2.56)$$

Now we will proceed to simplify the previous expression in the particular case of periodic geometries. For more generic geometries it is better to describe the losses of total pressure, i.e., of mechanical energy, in terms of loss coefficients instead of in terms of friction factors. We will see that in fact the losses of mechanical energy are directly related to entropy production. We will do this after discussing the case of periodic pipes.

### 2.5.4 Periodic Pipes

Expression (2.56) is valid for arbitrary axially symmetric geometries in horizontal position. However, in the particular case of periodic pipes, i.e., when there is a period $L$ such that $R_w(x) = R_w(x + L)$ for all $x$, the expression for the pressure drop (2.56) can be simplified greatly. The geometry configuration of a periodic pipe is depicted in Figure 2.3. Since the geometry is periodic and the flow fully developed, it is plausible to assume that the velocity field $u$ would be periodic as well, i.e., $u(r, 0) = u(r, L)$, where $L$ is the period of the pipe. From this it follows that the integral of $u_x^2$ over $\Gamma_{in}$ in the right hand side of (2.56), cancels the one over $\Gamma_{out}$.

After doing this, we are left with the following expression for the pressure drop over one period, i.e., from section $x = 0$ to $x = L$,

$$\Delta p = \frac{1}{\Delta p_x} \int_{\Gamma_{in}} pn_x dS - \frac{\mu}{\Delta p_x} \int_{\Gamma} \frac{\partial u_x}{\partial n} dS,$$  \hspace{1cm} (2.57)$$
2.6 Flow in Arbitrary Conduits

2.6.1 Loss Coefficients

In the previous section we derived an expression for the friction factor, describing the pressure losses in arbitrary horizontal periodic pipes. However, the losses of mechanical energy in internal flows also come from other kind of components, such as valves, fittings, bends, and elbows, and these components are not necessarily aligned in a horizontal position. These kind of components disturb the flow of the fluid and therefore cause additional losses. In a typical system with long pipes, these losses are minor compared to the total losses in the pipe lines (called major losses), that is why they are often named minor losses [15]. Even though this is generally true, sometimes this is not the case, for example, in systems with several turns and valves within short distance.

\[ n_x \] is the \( x \)-component of the normal vector to the surface, \( |\Gamma_{in}| = \pi R_w^2(0) \), and \( \Gamma \) is the wall of the pipe between \( x = 0 \) and \( x = L \). This formula also tells us that the pressure drop consists of two parts, one due to skin friction, \( \Delta p_S \), and one due to the pressure forces acting on the wall of the pipe, \( \Delta p_p \). In the particular case of a straight pipe, i.e., for Poiseuille flow, \( n_x = 0 \) and consequently (2.57) only contains the integral due to skin friction \( \Delta p_S \). After substituting the well-know parabolic profile for \( u_x \), we obtain the result (2.48) for the laminar friction factor in a straight pipe.

In order to be able to use (2.57) for computing the friction factor, we need to approximate the normal derivative \( \partial u_x / \partial n \), and the pressure \( p \) at the wall of the pipe. We can do this by solving numerically the Navier-Stokes equations (2.53), or by approximating this solution via an asymptotic method. We will discuss both approaches in Chapter 3. Now we turn our attention to the problem of describing the losses for more general components.
These losses are commonly introduced phenomenologically by adding a loss term to the Bernoulli’s equation [113]. Then, minor and major losses are described in terms of the loss coefficient $K_e$ [73],

$$K_e = \frac{2\varphi_\Omega}{u_m^2},$$  \hspace{1cm} (2.58)

where $\varphi_\Omega [\text{J kg}^{-1}]$ is the specific dissipation of mechanical energy due to the component over the domain $\Omega$, and $u_m$ is a characteristic mean velocity (for the situation described in Section 2.5.3, one can take $u_m = U_x$). Then the viscous dissipation $\varphi$ is added to the Bernoulli’s equation. For a component with inlet $\Gamma_{\text{in}}$ and outlet $\Gamma_{\text{out}}$ the Bernoulli’s equation with viscous dissipation reads [15]

$$\dot{m}\left(\alpha_{\text{in}}\frac{u_{\text{in}}^2}{2} + \frac{p_{\text{in}}}{\rho} + gx_{3,\text{in}}\right) + \dot{W}_{\text{pump}} = \dot{m}\left(\alpha_{\text{out}}\frac{u_{\text{out}}^2}{2} + \frac{p_{\text{out}}}{\rho} + gx_{3,\text{out}} + \varphi_\Omega\right),$$  \hspace{1cm} (2.59)

where $\dot{m} [\text{kg s}^{-1}]$ represents the mass flow rate, $\dot{W}_{\text{pump}} [\text{W}]$ the effective power input (due to a pump, compressor, etc.) and $\varphi_\Omega$ the specific dissipation within the domain $\Omega$. The terms $u_{\text{in},\text{r}}, p_{\text{in}}$ and $x_{3,\text{in},\text{r}}$ stand for the average velocity, pressure and the height (measured in the opposite direction of the acceleration $g$) at the inlet $\Gamma_{\text{in}}$. The variables with subscript $\text{out}$ describe the respective values at the outlet $\Gamma_{\text{out}}$.

The coefficients $\alpha_{\text{in}}$ and $\alpha_{\text{out}}$, are dimensionless kinetic energy correction factors. They take into account how the velocity is distributed over the cross section, a value of $\alpha = 2$ corresponds to a parabolic profile, and $\alpha = 1$ to a flat profile, for example. We will derive their general form later.

In the particular case of horizontal periodic pipes (also for straight pipes of course), the loss coefficient $K_e$ can be rewritten in terms of the friction factor $f$ by

$$K_e = f \frac{L}{D}, \text{ i.e., } f = \frac{\varphi}{2D} \frac{2D}{L u_m^2},$$  \hspace{1cm} (2.60)

where $L$ is the length (period) of the pipe and $u_m = U_x$ as in (2.52).

In order to be able to use (2.59) to describe the flow through a given component, we need to be able to determine the specific dissipation $\varphi$. For turbulent flows, loss coefficients for many type of components have been obtained experimentally and can be found in general fluid mechanics textbooks. These coefficients are handled as constant, which is only valid for highly turbulent flow, because for turbulent flow $\varphi \sim u_m^2$ [39]. For laminar flow conditions these coefficients are not commonly found and since under these conditions $\varphi \sim u_m$, the loss coefficients are expected to be dependent on the Reynolds number $Re$. Hence we are left with the problem to determine the specific dissipation. In order to do this, we need to relate the dissipation to a quantity that we can compute or
approximate, just as we did in the case of periodic pipes. We will do this by integrating the equation for the mechanical energy over a domain $\Omega$. At the same time, we will obtain a formal derivation for the extended Bernoulli’s equation (2.59).

### 2.6.2 Bernoulli’s Equation

Even though the losses in internal flows are commonly called pressure losses, they should be more accurately called losses of total head, because they occur when the total mechanical energy in the flow is dissipated by the action of viscosity. In order to see this, we first need to obtain the equation for mechanical energy. This can be done by taking the inner product of the momentum equations (2.3b) with $u$. By rewriting $u \cdot \frac{D}{Dt} u = D(\|u\|^2/2)/Dt$, using the fact that $u \cdot \nabla p = \nabla \cdot (pu) - p \nabla \cdot u = \nabla \cdot (pu)$ (were the second equality follows due to incompressibility), and making use of the identity $\nabla \cdot (\tau u) = \tau : \nabla u + u \cdot (\nabla \cdot \tau)$ (see Appendix A for more details), the equation for the mechanical energy takes the form

$$
\rho \frac{D}{Dt} \left( \frac{1}{2} |u|^2 \right) = -\nabla \cdot (pu) + \nabla \cdot (\tau u) - \tau : \nabla u + \rho u \cdot g. \tag{2.61}
$$

This equations shows that the convection of kinetic energy (left hand side term) is balanced by the sum of the work by pressure forces (first term right hand side.), work of viscous stresses (second term rhs.) and the work done by the body forces (fourth term rhs.), minus the viscous dissipation (third term rhs.) [76].

Before proceeding to integrate over the domain $\Omega$, we have to perform some preliminary calculations. First, we represent the volume force as the gradient of a potential, i.e., $g = -\nabla (gx_3)$, (where we have oriented $g = -ge_3$), and rewrite it in divergence form as follows

$$
u \cdot g = -g \nabla \cdot (x_3 u), \tag{2.62}
$$

where we used that the fluid is incompressible, i.e., $\nabla \cdot u = 0$. Now we proceed with the term describing the convection of kinetic energy. Since loss coefficients are defined for steady state conditions, we have that

$$
\frac{D}{Dt} \left( \frac{1}{2} |u|^2 \right) = u \cdot \nabla \left( \frac{1}{2} |u|^2 \right)
= \nabla \cdot \left( u \frac{1}{2} |u|^2 \right) - \frac{1}{2} |u|^2 \nabla \cdot u
= \nabla \cdot \left( u \frac{1}{2} |u|^2 \right), \tag{2.63}
$$

where the last equality again follows due to incompressibility. By substituting (2.62)
Figure 2.4: Three dimensional pipe with inlet $\Gamma_{\text{in}}$, outlet $\Gamma_{\text{out}}$. The wall of the pipe is denoted by the surface $\Gamma$, with normal vector $\mathbf{n}$. The potential energy is determined by the difference in height (in the opposite direction of gravity) between the inlet and the outlet, namely $x_{3,\text{in}} - x_{3,\text{out}}$.

and (2.63) into (2.61) the equation for mechanical energy takes the form

$$
\rho \nabla \cdot \left( \left( \frac{1}{2} |\mathbf{u}|^2 + \frac{p}{\rho} + gx_3 \right) \mathbf{u} \right) = \nabla \cdot (\tau \mathbf{u}) - \tau : \nabla \mathbf{u}.
$$

(2.64)

Finally, we can integrate this equation over the domain $\Omega$, see Figure 2.4. By using the divergence theorem we rewrite the left hand side volume integrals into surface integrals over $\Gamma \cup \Gamma_{\text{in}} \cup \Gamma_{\text{out}}$. The no-slip condition at the wall of the component $\mathbf{u} |_{\Gamma} \equiv \mathbf{0}$, implies that the integrals over $\Gamma$ are zero. Then grouping the integrals over $\Gamma_{\text{out}}$ on the left hand side and the ones over $\Gamma_{\text{in}}$ on the right hand side we obtain

$$
\rho \int_{\Gamma_{\text{out}}} \left( \frac{|\mathbf{u}|^2}{2} + \frac{p}{\rho} + gx_3 \right) \mathbf{u} \cdot \mathbf{n}_{\text{out}} dA + \int_{\Omega} \tau : \nabla \mathbf{u} dV = \\
\rho \int_{\Gamma_{\text{in}}} \left( \frac{|\mathbf{u}|^2}{2} + \frac{p}{\rho} + gx_3 \right) \mathbf{u} \cdot \mathbf{n}_{\text{in}} dA + \int_{\Omega} \nabla \cdot (\tau \mathbf{u}) dV,
$$

(2.65)

where we have used that at $\Gamma_{\text{in}}$, $\mathbf{n} = -\mathbf{n}_{\text{in}}$, and that at $\Gamma_{\text{out}}$, $\mathbf{n} = \mathbf{n}_{\text{out}}$. This integrated version of the equation for the mechanical energy now resembles the Bernoulli’s equation (2.59). However they are not exactly the same, this depends on the representative values which one considers at the inlet and at the outlet of the component. For the velocities, it is standard to take the cross-sectional average velocity $u_{\text{in}}$, and the kinetic energy correction factor $\alpha_{\text{in}}$, at the inlet $\Gamma_{\text{in}}$ of the component. These are defined as

$$
u_{\text{in}} := \frac{1}{|\Gamma_{\text{in}}|} \int_{\Gamma_{\text{in}}} \mathbf{u} \cdot \mathbf{n}_{\text{in}} dA \quad \text{and} \quad \alpha_{\text{in}} := \frac{\rho}{u_{\text{in}}^2} \int_{\Gamma_{\text{in}}} |\mathbf{u}|^2 \mathbf{u} \cdot \mathbf{n}_{\text{in}} dA,
$$

(2.66)
where \(|\Gamma_{in}|\) stands for the area of the inlet of the component and \(\dot{m}\) is the mass flow rate. In an analogous way, we can define the average velocity \(u_{in}\) and kinetic energy correction factor \(\alpha_{in}\), at the inlet of the component.

Now we come to the problem of choosing representative values for the pressure and the height. If we want to rewrite the integrated version of the equation for mechanical energy exactly in the form appearing in (2.59), we have to define the representative values of the pressure \(p_{in}\), and the height \(x_{3, in}\) as

\[
p_{in} := \frac{\rho}{\dot{m}} \int_{\Gamma_{in}} p u \cdot n_{in} dA \quad \text{and} \quad x_{3, in} := \frac{\rho}{\dot{m}} \int_{\Gamma_{in}} x_{3} u \cdot n_{in} dA.
\]  

(2.67)

Again, we define the respective values at the outlet \(\Gamma_{out}\), in an analogous way. Thus, substituting (2.66) and (2.67), into (2.65), and factorizing the mass flow rate \(\dot{m}\), which is defined as

\[
\dot{m} := \int_{\Gamma_{in}} \rho u \cdot n_{in} dA = \int_{\Gamma_{out}} \rho u \cdot n_{out} dA,
\]  

(2.68)

we finally obtain

\[
\dot{m} \left( \alpha_{in} \frac{u_{in}^2}{2} + \frac{p_{in}}{\rho} + gx_{3, in} \right) + \int_{\Omega} \nabla \cdot (\tau u) \, dV = \dot{m} \left( \alpha_{out} \frac{u_{out}^2}{2} + \frac{p_{out}}{\rho} + gx_{3, out} + \frac{1}{\dot{m}} \int_{\Omega} \tau : \nabla u \right).
\]  

(2.69)

We can now recognize the similarity between this equation and (2.59). The second term on the left hand side represents the work done by viscous stresses in the fluid against the surroundings to change the kinetic energy of the fluid, and it can be reinterpreted as a power input given by a pump for example.

The most important conclusion at which we arrive, by simply comparing (2.69) and (2.59), is that the specific dissipation of mechanical energy inside the domain \(\varphi_{\Omega}\), is given by

\[
\varphi_{\Omega} = \frac{1}{\dot{m}} \int_{\Omega} \tau : \nabla u \, dV = \frac{2\mu}{\dot{m}} \int_{\Omega} \left\| \frac{1}{2} \nabla u + \frac{1}{2} (\nabla u)^T \right\|^2 \, dV,
\]  

(2.70)

where \(\| \cdot \|\) denotes the Frobenius norm of a tensor (see (A.6)). The second equality above, is obtained after substituting the constitutive equation (2.6a) for \(\tau\), which yields (see
Governing Equations for Fluid Flow and Heat Transfer

Appendix A eq. (A.10) for more details)

\[ \tau : \nabla \mathbf{u} = 2\mu \| \mathbf{D} \|^{2} = 2\mu \left| \frac{1}{2} \nabla \mathbf{u} + \frac{1}{2} (\nabla \mathbf{u})^{T} \right|^{2} \geq 0, \quad (2.71) \]

where the inequality holds provided that the viscosity coefficient satisfies \( \mu > 0 \). This inequality expresses the experience that during deformation mechanical energy is dissipated into heat by the action of friction, and that dissipation is always positive. In fact, in the next section, we will see that the dissipation of mechanical energy from a thermodynamic point of view is directly linked to the production of entropy in a flow field.

Summarizing, formula (2.70) gives us an analytical expression for the specific viscous dissipation within any arbitrary component. This formula also tells us that the losses of mechanical energy occur locally in the presence of velocity gradients.

Unlike velocity or temperature, the dissipation of mechanical energy (entropy production) can not be measured directly. However, in a practical application, we could use (2.70) as an indirect way for measuring the dissipation of mechanical energy, for example, by estimating the gradients of velocity \[76\]. Under a computational setting, we could first solve for the velocity field \( \mathbf{u} \) in the domain of interest and integrate \( \tau : \nabla \mathbf{u} \) to obtain the overall dissipation inside the component. The main advantage of determining the loss coefficient by using the local viscous dissipation, is that it allows us to see where the losses of mechanical energy take place.

When using (2.70), one should keep in mind that \( \varphi_{\Omega} \) only represents the dissipation of mechanical energy which takes place inside the domain \( \Omega \). However the definition of the loss coefficient \( K_{e} \), given in (2.58) should include the losses of mechanical energy that take place inside, as well as upstream (before) and downstream (after) the component. This is specially important in the case of “short” components, such as bends, junctions, valves, etc., and in the case of large Reynolds numbers \[40\].

Regarding the definition for the representative pressure given in (2.67), we would like to point out that in many textbooks it is assumed that the pressure is constant over the cross-section, and that the difference in height along the cross section is negligible. Under these assumptions, the representative pressure \( p_{\text{in}} \) coincides with the constant value of the pressure, in (2.57) this assumption was used.

\[ \text{2.6.3 Entropy Production} \]

As we have seen, even though the losses in internal flows are commonly called pressure losses, they should be more accurately called losses of total pressure, because they occur
when the total pressure (the mechanical energy) is dissipated [41]. The dissipation of mechanical energy from a thermodynamic point of view is directly linked to the production of entropy in a flow field. This can be seen more clearly via the entropy equation (2.5), which without heat sources can be written as

\[
\rho \frac{Ds}{Dt} = \frac{1}{T} \tau : \nabla u - \frac{1}{T} \nabla \cdot q \\
= \frac{1}{T} \tau : \nabla u - \frac{q}{T^2} \nabla T + \frac{1}{T^2} \nabla T \cdot q - \frac{1}{T} \nabla \cdot q \\
= \frac{1}{T} \tau : \nabla u - \frac{q}{T^2} \nabla T - \nabla \cdot \left( \frac{1}{T} q \right),
\] (2.72)

where the first equality follows by adding and subtracting \( q \cdot \nabla T / T^2 \), and the second equality by using the chain and product rules for differentiation. In (2.72) the rate of change of entropy of a material particle can be split into two contributions, i.e., a rate of entropy production (always positive) with the value

\[
\rho \frac{Ds_{\text{irr}}}{Dt} = \frac{1}{T} \tau : \nabla u - \frac{q}{T^2} \nabla T \geq 0,
\] (2.73)

and an entropy flux vector \( q / T \), which can be positive, zero or negative

\[
\rho \frac{Ds_{\text{rev}}}{Dt} = -\nabla \cdot \left( \frac{1}{T} q \right),
\] (2.74)

where \( s = s_{\text{irr}} + s_{\text{rev}} \). The right hand side of (2.73), represents the irreversible actions of friction and heat conduction in the fluid particle. The second law of thermodynamics requires that the entropy production due to irreversible phenomena should be positive, this will give us conditions for the viscosity \( \mu \) and for the coefficient of heat conductivity \( \kappa \). When we substitute the constitutive equation (2.6a) for \( \tau \), we can see that (see Appendix A, eq. (A.10) for more details)

\[
\frac{1}{T} \tau : \nabla u = \frac{2\mu}{T} \|D\|^2 = \frac{2\mu}{T} \left\| \frac{1}{2} \nabla u + \frac{1}{2} (\nabla u)^T \right\|^2 \geq 0,
\] (2.75)

where the inequality holds provided that the viscosity \( \mu > 0 \). On the other hand, when we substitute the constitutive equation for the heat flux \( q \) (2.6b) we see that

\[
-\frac{q}{T^2} \nabla T = \frac{\kappa}{T^2} |\nabla T|^2 \geq 0,
\] (2.76)

where the inequality holds provided \( \kappa > 0 \). Hence the second law of thermodynamics is satisfied by taking positive values for the coefficients of viscosity and thermal conductivity.

The inequality (2.75) expresses the experience that during deformation mechanical en-
energy is dissipated into heat by the action of friction, but that heat cannot be changed into mechanical energy by the action of friction during deformation. The inequality (2.76) states that the heat flux vector must flow in the direction of decreasing temperature. The quantity appearing in (2.74) represents the change in entropy which the particle experiences from its neighborhood, because the divergence of the entropy flux is the difference between the inflowing and outflowing entropy flux [113].

As we can see from (2.73), the production of entropy occurs locally in the presence of velocity and temperature gradients, and it is represented by the right hand side in (2.73). Once the flow field \( u \) and temperature field \( T \) are known, we can compute the total entropy production by integrating the right hand side of (2.73) over the domain under consideration.

For a nearly isothermal process, the entropy production due to temperature gradients is negligible. The resulting form of the equation, then gives an expression in which the local entropy production is only due to the viscous dissipation contribution only, i.e.,

\[
\rho \frac{D S_{\text{vec}}}{D t} = 2\mu \frac{1}{T} \left\| \nabla u + \frac{1}{2} (\nabla u)^T \right\|^2.
\] (2.77)

In this case, we can link the entropy production with the specific viscous dissipation \( \varphi_{\Omega} \), which can also be interpreted as the power needed to transport the fluid through the system. Unlike velocity or temperature, the measurement of entropy cannot be performed directly. Instead, one can use the previous equation as an indirect way of characterizing the flow irreversibility [76].

This link allows us to have a thermodynamical interpretation of the losses of mechanical energy. In addition, it gives us the opportunity to see where the losses are more significant, hence providing us with a method to identify regions where a modification of the geometry could lead to less losses and therefore to significant improvement. In fact, the usage of a thermodynamic second law analysis is necessary in order to evaluate the efficiency of complex systems.
Chapter 3

Isothermal Laminar Flow

In this chapter we tackle the problem of laminar flow. We start by discussing the role of the Darcy friction factor in generating reduced models for flows in pipes. We do this numerically and also analytically. Then we present a two-dimensional finite element model, which allows us to solve the problem of fully developed flow in a corrugated pipe in an effective way. This is done by decomposing the pressure into a linear and oscillatory component, which allows us to numerically solve the equations on just one single period of the pipe, hence limiting computational costs. The presented numerical model can be used to predict the influence of wall-shape on the flow inside the corrugated pipe. As an example, we study the problem of finding an optimal wall-shape using this model.

In addition, we also derive an analytical formula for estimating the friction factor in axially symmetric pipes with arbitrary wall-shape. The formula is derived by combining an integral expression for the friction factor with asymptotic solutions to the Navier-Stokes equations derived via the method of slow variations. We evaluate the accuracy of the analytical formula for a set of sinusoidal pipes, where we consider three different parameters for this study, i.e., the amplitude and period of the corrugations and the Reynolds number. In the regions where the analytical formula is accurate, it proves to be the better alternative, because it is about $10^3$ times faster than the FEM numerical model. In the regions where the formula is not accurate, the numerical model presented in this section, is still an efficient way to compute the flow field and the friction factor. We conclude the chapter with a section on wall-shape design in which we address the problem of finding a wall-shape that maximizes the flow rate for a given pressure gradient. The material discussed in this chapter is an extended version of the work which we presented in [101, 103].
3.1 Introduction

As we discussed in Section 2.6.1, for practical applications, the losses of mechanical energy (total pressure losses), are expressed by a friction factor in the case of straight or periodic corrugated pipes, or by a loss coefficient for general components. The effect of wall-shape on the losses of mechanical energy is of interest in many areas, and applications range from microfluidics in biology [5] to large scale problems in the LNG industry [127].

The study of flow in non-straight pipes dates back to Nikuradse’s experiments [78], whose results obtained from artificially roughened pipes were later arranged in the well-known form of the Moody diagram [72], where for laminar flow, the friction factor is usually shown as independent of the roughness. Despite this, several numerical and experimental studies have shown that the contribution of wall-shape is not trivial, even in the laminar case. This happens specially when dealing with a specific wall-shape design, rather than just with some random roughness [9,47,59].

The studies on the effects of wall-shape on the pressure drop can be divided into three main categories: experimental studies [25,50,78], numerical methods [9,28,36,41,61,77,121], and analytical methods [10,12,18,53,109,125]. Deiber et al. conducted experiments and numerical simulations for flow through sinusoidal pipes [25]. Among the numerical investigations, there are papers based on the Stokes equations (creeping flow approximation) [36,121], and more recent papers which consider the full steady Navier-Stokes equations [41,77]. Analytical methods are specially useful for applications that require to compute the friction factor for many different wall-shapes, as for instance in wall-shape optimization or flow in large networks.

In this chapter we derive an analytical formula for estimating the friction factor in axially symmetric pipes of arbitrary shape. The formula is derived by using asymptotic solutions of the Navier-Stokes equations in primitive variables, obtained via the method of slow-variations (see [125] for an overview of the existing literature on this method). The key aspect in this derivation is that by combining the asymptotic solutions with an integral expression for the friction factor, it is possible to obtain very good approximations, even when considering only the leading order terms of the asymptotic solutions. In addition, we compare the estimates of the analytical formula, with a numerical model of the full steady Navier-Stokes equations. Based on the numerical model, we systematically evaluate the accuracy of the asymptotic formula, depending on three different parameters, the amplitude and period of the pipe, and the Reynolds number. The analytical formula allows us to compute the friction factor in a much faster way than with the numerical model. Our results are in good agreement with Sisavath et al. [109] where similar asymptotic expressions were derived using the stream function formulation. However, in order to be able to capture the dependency of the friction factor on
3.2 Governing Equations and Geometry

the period of the pipe, they reported necessary to take the asymptotic solutions up till second order. In our case, we show that by exploiting the integral formula of the friction factor, the effect of the amplitude can be captured, even when using only the leading order terms of the asymptotic solution. This is particularly advantageous in situations, where due to the nature of the problem, it is very complicated to obtain higher order terms of an asymptotic solution.

Besides this effort for obtaining an analytical formula, we also study the effects of wall-shape from the physical point of view. In particular, we address the problem of finding an optimal wall-shape, i.e., one that maximizes the flow rate for a given applied pressure gradient. We discuss the influence of the geometry and of the Reynolds number on the flow. We present the results for two essentially different configurations, namely the cavity configuration and the slowly varying configuration. In both cases we show that it is possible to increase the flow rate by a modification of the wall-shape. The situation looks especially promising for “slowly varying” geometry, and from our numerical simulations an increment of up to 120% in flow rate was observed. When dealing with large pressure gradients, we show that an asymmetric (in the axial direction) wall-shape can perform better than the symmetric counterpart. We study the dependency of the optimal ratio of expansion and contraction on the pressure gradient.

3.2 Governing Equations and Geometry

We consider the incompressible Navier-Stokes equations, i.e., equations (2.37a) and (2.37b). For steady, axially symmetric, laminar flow in cylindrical coordinates, these equations take the following form

\[ u_x \partial_x u_x + u_r \partial_r u_x = \nu \left( \partial_{xx} u_x + \partial_{rr} u_x + \frac{1}{r} \partial_r u_x \right) - \frac{1}{\rho} \partial_x p, \quad (3.1a) \]

\[ u_x \partial_x u_r + u_r \partial_r u_r = \nu \left( \partial_{xx} u_r + \partial_{rr} u_r + \frac{1}{r} \partial_r u_r - \frac{1}{r^2} u_r \right) - \frac{1}{\rho} \partial_r p, \quad (3.1b) \]

\[ \partial_x u_x + \partial_r u_r + \frac{1}{r} u_r = 0, \quad (3.1c) \]

where the corresponding variables are the axial coordinate \( x \), the radial coordinate \( r \), the axial velocity \( u_x \), the radial velocity \( u_r \), and the pressure \( p \). The constants \( \nu \) and \( \rho \) represents the kinematic viscosity and the density of the fluid, respectively. The angular component does not play a role due to the assumption of axially symmetric flow.

The geometry under consideration is an axially symmetric pipe, depicted as in Figure 3.1. As we discussed in Section 2.5.2, the location of the wall of the pipe can be described in terms of the cylindrical basis vectors \( e_r, e_\phi, e_z \), via the parametrization
Figure 3.1: Axisymmetric pipe with center line along the x-axis. Γ stands for the wall of the pipe, Γ_{in} for the cross section at the inlet of the pipe (x = 0) and Γ_{out} for the cross section at the outlet (x = L_p).

\[ x(\phi, x) = R_w(x)e_r + xe_x, \] with \( 0 \leq \phi < 2\pi, 0 \leq x \leq L_p \). The outer unit normal vector \( n \) (2.50a), and the surface element \( dS \) (2.50b) can be expressed in terms of the function \( R_w(x) \) describing the radius of the pipe. For the sake of convenience, we recall these expressions here

\[ n = \frac{e_r - R_w'(x)e_x}{\sqrt{1 + R_w'(x)^2}}, \] \hspace{1cm} (3.2a)

\[ dS = R_w(x)\sqrt{1 + R_w'(x)^2}d\phi dx, \] \hspace{1cm} (3.2b)

where \( (\cdot)' \) denotes the derivative with respect to the axial variable x. At the wall Γ of the corrugated pipe, we consider the no-slip boundary condition (2.44), which for this particular geometry reads

\[ u(R_w(x), x) = 0, \] \hspace{1cm} for \( 0 \leq x \leq L \). \hspace{1cm} (3.3)

In addition, one should prescribe boundary conditions at the inlet Γ_{in} and outlet Γ_{out}. For the numerical model we will impose periodic boundary conditions, while for the asymptotic model, we will prescribe a constant flow rate. We will discuss each case in the following sections.

### 3.2.1 The Darcy Friction Factor

As we discussed in Section 2.5.1 and 2.6.1, a very important quantity in the analysis of pipe flow is the so-called pressure drop, or to be more precise, the losses of mechanical energy. The losses of mechanical energy are usually described in terms of the mean flow rate, and they also determine the power requirements of the device to maintain the flow. For fully developed horizontal flow in corrugated (also in straight) pipes, we saw that
the pressure drop (or losses of mechanical energy) along one period, i.e., from \( x = 0 \) to \( x = L \), can be expressed in terms of the Darcy friction factor \( f \) as (see (2.47))

\[
\Delta p = f \frac{L \rho U_x^2}{D},
\]  

(3.4)

where \( D = 2R_w(0) \) is the reference (minimum inner) diameter of the pipe, \( \rho \) is the density and \( U_x \) the average velocity over the cross section of the pipe. Under the assumption of horizontal fully developed flow, we saw that the pressure drop \( \Delta p \) in (3.4), can be obtained from (see (2.57))

\[
\Delta p = \frac{1}{\Gamma_{in}} \int_{\Gamma} p n_x dS - \frac{\mu}{\Gamma_{in}} \int_{\Gamma} \frac{\partial u_x}{\partial n} dS,
\]  

(3.5)

where \( n = n_r e_r + n_x e_x \) denotes the normal vector to the surface \( \Gamma, |\Gamma_{in}| = \pi R_w^2(0) \), and \( \Gamma \) denotes the wall of the pipe between the sections \( x = 0 \) and \( x = L \).

We recall that (3.5) is suitable for describing the losses of mechanical energy for horizontal and periodic components. Under this setting, the losses of mechanical energy coincide with the pressure drop along the pipe. However in a more general case, for non-periodic components for example, the losses of mechanical energy are not given by the pressure drop only. In Section 2.6.2 we saw that in the most general case, the losses of mechanical energy can be expressed in terms of the loss coefficient \( K_e \), in combination with (2.59), which is an extension of Bernoulli’s equation. The specific dissipation of mechanical energy \( \varphi_\Omega \) appearing in this equation is directly linked to entropy production, and it can be computed with (2.70), which reads

\[
\varphi_\Omega = \frac{2\mu}{m} \int \left\| \frac{1}{2} \nabla u + \frac{1}{2} (\nabla u)^T \right\|^2 dV.
\]  

(3.6)

Expression (3.6) allows us to compute the losses of mechanical energy in the most general case. In any case, both (3.5) and (3.6) require the velocity field \( u \) as input for computing the the friction factor (or loss coefficient). We will now show how these formulas can be applied in the simple case of fully developed horizontal pressure driven flow in a straight pipe.

In the case of a straight pipe the radius is constant, i.e., \( R_w \equiv R_w(0) \). Since the flow is assumed steady and horizontal we have \( \partial_t(,.) = 0 \) and \( u_r \equiv 0 \). Therefore the continuity equation (3.1c) simplifies to

\[ \partial_x u_x = 0, \]

which implies that \( u_x \) is independent of the axial coordinate \( x \), i.e., \( u_x = u_x(r) \). Substi-
Solving this into the axial momentum equation (3.1a), we obtain
\[ \nu \left( \partial_r u_x + \frac{1}{r} \partial_r r u_x \right) - \frac{1}{\rho} \partial_x p = 0. \]

Denoting the constant applied pressure gradient as \( \beta := -\partial_x p = \Delta p / L \), using the identity \( \frac{1}{r} \partial_r \left( r \partial_r u_x \right) = \partial_r u_x + \frac{1}{r} \partial_r r u_x \), and integrating with respect to \( r \) we obtain
\[ r \partial_r u_x = -\beta r^2 + c_1(x), \]
where \( c_1 \) is the integration constant (possibly a function of \( x \)), however evaluating the previous equation at \( r = 0 \), we can see that \( c_1 = 0 \). An alternative way of interpreting this boundary condition is that \( u_x \) should remain finite at the centerline of the pipe.

Now dividing by \( r \) and integrating with respect to \( r \) once more we obtain
\[ u_x = -\beta r^2 + c_2(x), \]
where \( c_2 \) is another integration "constant". Using the no-slip boundary condition (3.3), we have \( u_x(R_w) = 0 \) and we obtain the parabolic velocity profile
\[ u_x(r) = \frac{\beta}{4\mu} \left( R_w^2 - r^2 \right), \quad (3.7) \]
where \( R_w \) is the radius of the pipe and \( \beta = -\partial_x p \) is the applied pressure gradient. Now we can compute the volume flow rate in a straight pipe \( Q_{sp} \) (and consequently the average axial velocity \( U_x \)) by integrating the velocity profile over the cross section, this yields
\[ Q_{sp} := \int_{r_{in}} u_x dA = 2\pi \int_0^{R_w} u_x r dr = \frac{\pi \beta R_w^4}{8\mu} \quad \text{and} \quad U_x := \frac{Q_{sp}}{\pi R_w^2}. \quad (3.8) \]

Now we will show how we can use (3.5) for computing the pressure loss. Since we consider a straight pipe, we have \( R'_w \equiv 0 \). This implies (see (3.2a),(3.2b)) that \( n = e_r \) and \( dS = R_w d\phi dx \). Therefore (3.5) simplifies to
\[ \Delta p = -\frac{\mu}{\pi R_w^2} \int_0^L \int_0^{2\pi} \left| \partial_r u_x \right|_{r=R_w} R_w d\phi dx = \beta L, \]
which confirms that \( \Delta p = \beta L \). Now we can substitute the average axial velocity \( U_x \), and the pressure drop \( \Delta p \) in (3.4) to recover the well-known relationship for the friction factor of laminar flow in a straight pipe \( f_{sp} \) [15]
\[ f_{sp} = \frac{64}{Re} \quad \text{where} \quad Re = \frac{U_x 2R_w}{\nu}. \quad (3.9) \]
If we use (3.6) to compute the friction factor, we recover the same result. Nonetheless, it is important to keep in mind that (3.6) has the advantage that is also applicable for arbitrary three-dimensional geometries. Once the friction factor (or the loss coefficient in the more general case) is known, equation (3.4) ((2.59) in the general case) can be used as a lumped model for describing the flow inside any kind of component. The main difficulty is to efficiently determine the friction factor (loss coefficient) that correctly describes the losses of mechanical energy due to the component. Formulas (3.5) and (3.6) allow us to compute the friction factor, however, in order to use them, we require the velocity field $u$. We will proceed in two ways, first we will discuss how to address this problem with a numerical model, and then we will derive and asymptotic formula for computing the friction factor in a very efficient way. We will compare both alternatives and will discuss in which cases it is convenient to use each of the alternatives.

### 3.3 Numerical Model

Now we proceed with the numerical alternative for computing the friction factor of fully developed flow in corrugated pipes. As we discussed in Section 2.4.3, the system of equations (3.1) needs to be provided with appropriate boundary conditions. A typical approach to model flow in a straight pipe [15], is to consider a prescribed velocity at the inlet $\Gamma_{in}$, the no-slip boundary condition (2.44) at the wall of the pipe, and the natural boundary condition (2.45), at the outlet of the pipe $\Gamma_{out}$. In Figure 3.2, the flow field which results from such boundary conditions has been sketched in the case of laminar flow in a straight pipe.

Due to the no-slip condition, the fluid particles which come in touch with the surface of the pipe come to a complete stop. This layer of still particles causes the neighboring layers of fluid to slow down gradually due to the action of viscosity, which gives rise to a boundary layer. The thickness of the boundary layer increases in the flow direction along the so-called hydrodynamic entrance region. The region located after the entrance region is called the hydrodynamic fully developed region. In the fully developed region, the velocity profile no longer changes in the flow direction, hence the name fully developed. The length of the entrance region is called the entry length. In practice, this length is usually taken to be the distance from the pipe entrance to the point where the wall shear stress (and thus also the friction factor) reaches within about 2 percent the fully developed value [15]. The hydrodynamic entry length is given approximately as $0.05 \text{Re} D$ for laminar flows [52]. In the case of turbulent flow, the fully developed region is defined in terms of time averaged quantities, i.e., the flow is fully developed when the time averaged velocity profile no longer changes in the flow direction. The hydrodynamic entry length for turbulent flow can be approximated by $1.359 \text{Re}^{1/4}$ [7].

In the case of a corrugated pipe, we can no longer expect that the velocity field will not
change in the axial direction. This is a consequence of the variations of the radius along the axial direction. However, in the laminar case, it is plausible to assume that at certain distance from the inlet, the velocity field will be periodic. This means that for fully developed flow in a periodic pipe, it is possible to reduce the computational domain to just one single period, which is very advantageous. To justify this assumption, we will perform some simulations in a geometry containing several periods, and will confirm that for laminar flow under steady state conditions, the period of the flow field indeed coincides with the period of the geometry.

The previous discussion justifies the periodicity of the velocity field. The pressure however, it is certainly not periodic, otherwise there would be no net flow in the axial direction. The periodicity in this case rather means that the losses of mechanical energy in each period should be the same. In a horizontal pipe this means that the pressure drop per period should be constant. Such a situation can readily be implemented using the periodic boundary conditions \(2.46\). However we will proceed in a slightly different manner.

Even though the pressure is not periodic we can still decompose the pressure as follows

\[
p(x, r) = \tilde{p}(x, r) - \beta x,
\]

where \(\tilde{p}(x, r)\) represents the pressure deviations due to the presence of the corrugation, and \(\beta [\text{Pa m}^{-1}]\) represents the average pressure gradient along the axial direction.

This transformation has been frequently used in the literature \([82, 88, 115, 124]\), and the main advantage of this reformulation is that the pressure fluctuation \(\tilde{p}\) is periodic, thus allowing us to reduce the computational domain to just one period. Thus, in our numerical model, we solve for the pressure fluctuation \(\tilde{p}\), instead of for the original pressure \(p\). Substituting (3.10) into (3.1a) the momentum equations take the form

\[
u \cdot \nabla u = \nu \nabla^2 u - \frac{1}{\rho} \nabla \tilde{p} + \frac{\beta}{\rho} e_x,
\]

Figure 3.2: Developing velocity profile in a straight pipe. The think lines show the boundary layer, which grows until it fills the whole cross-section of the pipe in the fully developed region.
where \( \beta e_x \) is a volume force term. Thus the system of equations which we solve numerically is

\[
\begin{align*}
\partial_x u_x + u_r \partial_r u_x &= \nu \left( \partial_{xx} u_x + \partial_{rr} u_x + \frac{1}{r} \partial_r u_x \right) - \frac{1}{\rho} \partial_x \tilde{p} + \frac{\beta}{\rho}, \\
\partial_x u_r + u_r \partial_r u_r &= \nu \left( \partial_{xx} u_r + \partial_{rr} u_r + \frac{1}{r} \partial_r u_r - \frac{1}{r^2} u_r \right) - \frac{1}{\rho} \partial_r \tilde{p}, \\
\partial_x u_x + \partial_r u_r + \frac{1}{r} u_r &= 0,
\end{align*}
\]

subject to the boundary conditions

\[
\begin{align*}
\text{No-slip:} \quad u(R_w(x), x) &= 0 \quad \text{for} \quad 0 \leq x \leq L, \\
\text{Periodicity:} \quad u(r, 0) &= u(r, L), \\
\tilde{p}(r, 0) &= \tilde{p}(r, 0),
\end{align*}
\]

where \( L \) denotes the period of the pipe.

The implementation works in the following way. First, we prescribe a total pressure drop of \( \Delta p = \beta L \) in one periodic section of the pipe. Second, we include the mean pressure gradient as the force term \( \beta / \rho \) in the Navier-Stokes equations (3.12), with variables \( u_x, u_r \) and \( \tilde{p} \), and solve these equations numerically. Finally, we compute the average axial velocity \( U_x \) and the \textit{volumetric flow rate} \( Q \), by integrating the axial velocity component \( u_x \) over the inlet of the pipe \( \Gamma_{in} \), which allows us to compute the resulting Reynolds number \( \text{Re} \), and friction factor \( f \), according to

\[
Q := \int_{\Gamma_{in}} u_x \, dA, \quad U_x := \frac{4Q}{\pi D^2}, \quad \text{Re} = \frac{U_x D}{\nu}, \quad f = \beta \frac{2D}{\rho U_x^2},
\]

where \( D = 2R_w(0) \) and \( R_w(0) \) corresponds to the smallest radius of the corrugated pipe.

The Navier-Stokes equations are solved with a mixed finite element model, with Lagrange \( P_2, P_1 \)-elements, i.e., the velocity is approximated with quadratic elements, and the pressure with linear elements. The order of approximation of the pressure is chosen to be one order less than for the velocity, in order to avoid an overdetermined system of equations [97]. The model used for these simulations, was implemented with the code Comsol Multiphysics [22].

The mesh used for discretizing the geometries studied in this section, is a mapped-mesh of quadrilateral elements, see Figure 3.3. The distribution of the elements in the center line is uniform, while in the radial direction, the density of the elements increases quadratically when approaching the wall. The mesh is designed to be finer close to the wall, in order to capture the strong gradients of the velocity field near the corrugated wall. The mesh was chosen to be coarser close to the center line, in order to reduce the
calculation time.

Figure 3.3: Mapped mesh for a sinusoidal pipe with 12 elements in the axial direction \( x \), and 10 elements in the radial direction.

Figure 3.4: Relative changes in the computed Reynolds number \( Re \) and friction factor \( f \), for two different pressure gradients \( \beta = 10^4, 1 \).

It is also appropriate to mention, that by using a mapped mesh (i.e., a quadrilateral mesh mapped into the sinusoidal geometry), it is possible to spare some computational time in mesh generation. For example, when one uses a boundary layer type mesh (which consists of a mesh of quadrilateral elements close to the boundary, and a triangular mesh in the core region), one needs to spend certain time in the generation of the triangular mesh.

3.3.1 Mesh Independence

In order to obtain grid independent solutions, the calculations were performed for different mapped meshes, until a new refinement provided no significant changes in the solution nor in the variables of interest, namely \( Re, Q, \) and \( f \). In Figure 3.4 we can observe the absolute value of the relative variations of the computed Reynolds number \( Re \), and friction factor \( f \), versus the total number of elements in the mesh. Notice that due to (3.14) an accurate value for \( Re \) implies an accurate value for \( Q \), and therefore we do not need to check again for its accuracy. For the construction of this plot, the initial number of elements in the axial and in the radial directions were 12 and 10, respectively, and subsequently, the number of elements in each direction was increased linearly until a further increase produced a change of less than 0.01% for the Reynolds number \( Re \), and the friction factor \( f \). For the case of a pressure gradient \( \beta = 1 \) (triangle markers), a variation of less than 0.01% in the Reynolds number \( Re \) is attained for a mesh with 48 elements in the axial direction and 40 elements in the radial direction, delivering a
3.3 Numerical Model

converged value of $Re \approx 0.6807$. On the other hand, the friction factor $f$ requires a mesh with $60 \times 50$ elements to present a variation of less than 0.01%, and converges to the value $f = 34.5304$. The number of elements increases with the mean pressure gradient $\beta$, and in order to obtain convergence for the case $\beta = 10^4$, we need a mesh with $84 \times 70$ elements. Of course, depending on the complexity of the geometry, the amount of elements required to achieve mesh independence increases. This is specially the case for the cavity type configuration (see Section 3.5), which basically are shapes with fast variations in the axial direction. Still mesh independence is achieved at lower costs than when using a boundary layer mesh. In order to warrant grid independence for all geometries, we implemented a routine which recursively refines the mapped mesh, until the relative change in the computed friction factor was less than 0.01%. Once this condition is met, the Reynolds number (and consequently the flow rate), automatically satisfy the same condition, as it can be observed in Figure 3.4.

The discretized nonlinear system of equations is solved using Newton iteration in combination with a zeroth order continuation for the mean pressure gradient $\beta$. This means that the solution computed at a given pressure gradient, is used as initial guess for the Newton iteration, at the next (higher) pressure gradient [97].

3.3.2 Validation

Since it is always instructive to compare results with experimental data, in Figure 3.5 we show a plot of experimental data (blue dots) obtained by Deiber et al. [25, p. 642], and the corresponding values obtained with the numerical model described above (red triangles). The geometry used in the experiments can be described by (after scaling with the minimum radius)

$$R^*_w(x^*) = 1 + \frac{a}{2} \left(1 + \sin \left(\frac{2\pi x^*}{L} - \frac{\pi}{2}\right)\right), \quad (3.15)$$

where $x^*$ and $R^*_w$ are the dimensionless axial coordinate and radius, respectively. The amplitude of the pipe is $a = 0.8571$ and the period $L = 8.9714$. The additional straight line in the plot, was added in order to show that, starting from a certain value for $Re$, the friction factor deviates from a straight line. This is in agreement with the conclusions of other authors [25,31].

The numerical model is able to accurately predict the friction factor up till $Re \approx 750$, where a change towards turbulence can be observed from the experimental data. This point has been signaled with a "T" in Figure 3.5. For $Re$ beyond this value, the steady state numerical model can not be expected to correctly describe the friction factor.

In order to have a better picture of the effect of the Reynolds number on the flow, in
Figure 3.5: Experimental results (blue dots) for the friction factor by Deiber et al. [25, p. 642], and values obtained with FEM simulations (red line), as function of the Reynolds number, for a sinusoidal pipe with parameters $a = 0.8571$ and $L = 8.9714$. All the values were converted from the variables used in Deiber et al. [25], to the ones used here.

Figure 3.6: Pressure fluctuations $\tilde{p}$, and streamlines for a sinusoidal pipe with radius at inlet $R_w(0) = 1$, amplitude $a = 0.8571$, period $L = 8.9714$, and various Reynolds numbers.
3.4 An Asymptotic Formula for the Friction Factor

Figure 3.6 we present plots of the fluctuation of the pressure $\hat{p}$, and the streamlines for this geometry, at different values of $Re$. The center line is located at $r = 0$, the wall of the pipe appears on the right hand side of the image, and the flow direction is upwards. For $Re = 36.15$ (Figure 3.6(a)), we can observe how the streamlines simply follow the wall of the pipe, and the pressure is almost constant along the cross sections. In Figure 3.6(b), at $Re = 81.33$, we can see indicated by an arrow, the onset of a vortex in the protrusion of the pipe, which is also signaled in Figure 3.5 with “V”. This vortex increases in size with $Re$, and in Figure 3.6(c), we can observe how the vortex almost fills the whole protrusion of the pipe, and how the pressure starts to be non-constant over cross sections, for instance at the cross section $x/R_w(0) = 7$. Although the experimental results at $Re = 927.7$ already exhibit a transition towards turbulence, it is still instructive to have a look at the results from the steady state numerical model. In Figure 3.6(d), we can observe how in this situation, the vortex completely fills the protrusion, and how it is squeezed by the core flow. As a result, the center of the vortex (signaled by arrows in Figure 3.6) migrates further in the flow direction, giving rise to a stagnation point. This can be observed from the variations of the pressure over cross sections.

From the discussion above, we see that the presence of a corrugated wall, can have a big influence on the flow field, and consequently on the friction factor as well. In fact, when compared to the friction factor of a straight pipe with radius $1$ (dashed line in Figure 3.5), the friction factor of this wavy pipe, is about 50%, and 30% smaller, at $Re \approx 81.33$, and $Re = 612.7$, respectively. This suggest that it should be possible to obtain a reduction in the friction factor.

Later in this chapter, we will address this question and will show that in fact it is possible to reduce the friction by correctly designing the wall-shape, but before that we will present the derivation of an asymptotic formula for computing the friction factor. Even though the numerical model presented in this section provides accurate results within reasonable time. It is convenient to have more efficient techniques to calculate the friction factor. Efficiency can be specially important in applications where one needs to analyze many different wall-shapes.

### 3.4 An Asymptotic Formula for the Friction Factor

In this section we derive an asymptotic formula for the friction factor in axially symmetric pipes of arbitrary shape. The formula is derived by using asymptotic solutions of the Navier-Stokes equations in primitive variables, obtained via the method of slow-variations (see [125] for an overview of the existing literature on this method). The key aspect in this derivation, is that by combining the asymptotic solutions with an integral expression for the friction factor, it is possible to obtain very good approximations, even when considering only the leading order terms of the asymptotic solutions. In addition,
we compare the estimates of the analytical formula with a numerical model of the full steady Navier-Stokes equations. Based on the numerical model, we systematically evaluate the accuracy of the asymptotic formula, depending on three different parameters, namely, the amplitude and period of the pipe, and the Reynolds number. The analytical formula allows us to compute the friction factor in a much faster way than the numerical model.

In Section 2.5.4, we saw that in the case of horizontal flow in axially symmetric periodic pipes, it is possible to write the pressure drop $\Delta p$ in terms of surface integrals over the wall of the pipe $\Gamma$ (2.57). In this formula the pressure losses appear as the sum of $\Delta p_S$ and $\Delta p_P$, which represent the pressure losses due to friction and to pressure forces at the wall respectively.

In order to be able to use (2.57) for computing the friction factor, we need to approximate the normal derivative $\partial_n u_x$, and the pressure $p$ at the wall of the pipe. We will do this by using the method of slow variations. With this asymptotic solution we will derive an integral expression for the friction factor. Since this expression only requires numerical integration in one single dimension, the method is very efficient. We will also systematically study the range of validity of the derived formula.

### 3.4.1 Method of Slow Variations

The method of slow variations exploits the geometric characteristics of boundaries that vary more slowly in some direction than others. The key idea of the method is to rescale the geometry in such a way that the variations become of the same order. This crucial step, enables us to take a geometrical parameter and transfer it as a coefficient in the scaled equations, which allows us to write the solution as an asymptotic expansion. One of the remarkable properties of the method is that it can handle arbitrarily large variations, provided that they take place slowly [125].

Asymptotic solutions for flow in axially symmetric pipes have been derived in several papers [53, 62, 125]. The derivation we present here follows the line of the paper by Kotorynski [53]. Before starting with the method of slow variations, we need to rewrite the Navier-Stokes equations (3.1) in dimensionless form as in Section 2.4, by defining the following variables

$$ u^*_x = \frac{u_x}{U_x}, \quad u^*_r = \frac{u_r}{U_x}, \quad x^* = \frac{x}{D}, \quad r^* = \frac{r}{D}, \quad p^* = \frac{p}{\rho U_x^2}. $$

(3.16)
Substituting these variables in (3.1) and applying the chain rule we obtain

\[
\text{Re} \left( u^*_x \partial_x u^*_x + u^*_r \partial_r u^*_x \right) = \partial_x u^*_x + \partial_r u^*_x + \frac{1}{r} \partial_r u^*_x - \text{Re} \partial_x p^*, \quad (3.17a)
\]

\[
\text{Re} \left( u^*_x \partial_x u^*_r + u^*_r \partial_r u^*_r \right) = \partial_x u^*_x + \partial_r u^*_r + \frac{1}{r} \partial_r u^*_r - \frac{1}{r^2} \partial_r u^*_r - \text{Re} \partial_r p^*, \quad (3.17b)
\]

\[
\partial_x u^*_x + \partial_r u^*_r + \frac{1}{r} \partial_r u^*_r = 0. \quad (3.17c)
\]

### 3.4.2 Reformulation in Slowly Varying Variables

Now we proceed to rescale (3.17), by using the assumption that the radius of the pipe varies slowly in the axial direction. This means that the dimensionless radius of the pipe \( R_w^*(x^*) \) can be written as

\[
R_w^*(x^*) = \hat{R}_w (\epsilon x^*) ,
\]

where \( \epsilon \) is a small dimensionless parameter characterizing the slow variation of the radius in the axial direction. This parameter can be taken directly from the expression for the radius, if available, for example, if the pipe radius would be of the form \( R_w(x) = (1 + \epsilon^2 x^2)^{1/2} \). In the case of a periodic pipe, one can consider the maximum variation of the radius \( a_c \) and compare it to the period of the pipe \( L \), i.e., we define \( \epsilon := a_c / L \). Relation (3.18) conveniently represents the assumption of slow variation because we have \( \partial_x, R_w^* = \epsilon \partial_x \hat{R}_w = \mathcal{O}(\epsilon) \). After this, we can apply a proper scaling, to obtain a domain in which the period is comparable to the variation of the radius. Formally this is done by defining the \textit{slowly varying variables}

\[
\hat{x} = \epsilon x^*, \quad \hat{r} = r^*, \quad \hat{u}_x = u^*_x, \quad \epsilon \hat{u}_r = u^*_r, \quad \epsilon^{-1} \hat{p} = p^* .
\]

Notice that in (3.18) we in fact assumed \( \hat{R}_w \) to be a function of the slowly varying variable \( \hat{x} \). Substituting these variables in (3.17) and multiplying the second and third equations by \( \epsilon \) and \( \epsilon^{-1} \), respectively, we obtain

\[
\epsilon \text{Re} \left( \hat{u}_x \partial_x \hat{u}_x + \hat{u}_r \partial_r \hat{u}_x \right) = \epsilon^2 \partial_{xx} \hat{u}_x + \partial_{xr} \hat{u}_x + \frac{1}{r} \partial_r \hat{u}_x - \text{Re} \partial_x \hat{p}, \quad (3.20a)
\]

\[
\epsilon^3 \text{Re} \left( \hat{u}_x \partial_x \hat{u}_r + \hat{u}_r \partial_r \hat{u}_r \right) = \epsilon^4 \partial_{xx} \hat{u}_r + \epsilon^2 \left( \partial_{xr} \hat{u}_r + \frac{1}{r} \partial_r \hat{u}_r - \frac{1}{r^2} \partial_{rr} \hat{u}_r \right) - \text{Re} \partial_r \hat{p}, \quad (3.20b)
\]

\[
\partial_x \hat{u}_x + \partial_r \hat{u}_r + \frac{\hat{u}_r}{\hat{r}} = 0. \quad (3.20c)
\]

This scaling transfers the parameter \( \epsilon \) from the geometry into the equation, where it appears as a coefficient. This allows us to vary this parameter, while keeping the domain fixed, namely \( 0 \leq \hat{x} \leq a / D \), and, \( 0 \leq \hat{r} \leq \hat{R}_w(\hat{x}) \). Formally, this means that we can write an \textit{asymptotic expansion} with respect to the parameter \( \epsilon \) for \( \hat{u}_x, \hat{u}_r, \) and \( \hat{p} \) in (3.20), i.e.,
where \( \hat{u}_{i,x} \), \( \hat{u}_{i,r} \) and \( \hat{p}_i \) denote the \( i \)-th term of the asymptotic expansion for the axial velocity, the radial velocity, and the pressure respectively. In particular we have \( \hat{u}_i = \hat{u}_{i,x} e_x + \hat{u}_{i,r} e_r \). Substituting these expressions into (3.20), and grouping the variables with respect to their order in \( \epsilon \), we obtain a set of equations for each of the orders in the asymptotic expansion. The no-slip boundary condition at the wall \( \Gamma \) for the resulting systems becomes

\[
\hat{u}_{i,x}(\hat{x}, \hat{r}; \epsilon) = \hat{u}_{i,r}(\hat{x}, \hat{r}; \epsilon) = \hat{p}(\hat{x}, \hat{r}; \epsilon) = \hat{p}_i(\hat{x}, \hat{r}) \epsilon^i,
\]

(3.21)

Boundary conditions at the inlet and outlet of the pipe do not need to be included. Instead, we have to prescribe conditions for the flow rate. This happens because \( \hat{u}_{i,x}, \hat{u}_{i,r}, \) and \( \hat{p} \), are assumed to be dependent only on the slow variables \( \hat{x} \) and \( \hat{r} \). Due to this assumption, the solution is valid in all regions, provided that we stay away from an inlet or outlet with imposed velocity (an inlet for example) [64]. In practical terms, this simply means that the solution is expected to be valid in the fully developed region. Hence we assume the flow rate \( Q \) to be constant, due to the continuity equation (3.1c), \( Q \) is independent of \( x \) and hence can be defined as

\[
Q := \int_{\Gamma_{in}} u_x \, dA = 2\pi \int_0^{R_w(0)} ru_x(x, r) \, dr.
\]

(3.23)

In an analogous way, we define the dimensionless flow rate \( \hat{Q} \). As we did with the velocity and pressure fields \( \hat{u} \) and \( \hat{p} \), we now expand the dimensionless flux as \( \hat{Q} = \hat{Q}_0 + \epsilon \hat{Q} + \epsilon^2 \hat{Q}_2 + ... \), where the scaled flow rates \( \hat{Q}_i \) are defined as

\[
\hat{Q}_i := 2\pi \int_0^{R_w(0)} \hat{r} \hat{u}_{i,x}(\hat{x}, \hat{r}) \, d\hat{r}.
\]

(3.24)

Since \( \hat{Q} = \hat{Q}_0 + \epsilon \hat{Q} + \epsilon^2 \hat{Q}_2 + ... \) must hold for arbitrary \( \epsilon \), it follows that

\[
\hat{Q}_0 = \hat{Q}, \quad \hat{Q}_i = 0 \text{ for } i = 1, 2, 3, \ldots
\]

(3.25)

Furthermore, using that \( \hat{u}_x(\hat{r}, \hat{x}) = u_x(r, x)/(U_x) \), the scaled flux can be written as

\[
\hat{Q}_0 = \hat{Q} = 2\pi \int_0^{R_w(0)} \hat{r} \hat{u}_x(\hat{x}, \hat{r}) \, d\hat{r} = \frac{2\pi}{U_x} \int_0^{R_w(0)} \hat{r} u_x(0, D\hat{r}) \, d\hat{r}.
\]

(3.26)

Substituting \( U_x \) from (3.14), and making a change of variables \( \hat{r} = \eta/D \), we get

\[
\hat{Q}_0 = \frac{\pi R_w^2(0)}{\int_0^{R_w(0)} ru_x(0, r) \, dr} \int_0^{D R_w(0)} \frac{\eta}{D^2} u_x(0, \eta) \, d\eta = \frac{\pi}{4}.
\]

(3.27)
Thus with our particular scaling, the flow rate $\hat{Q}$ is always $\pi/4$. Now we proceed to obtain the leading order solution of (3.20).

### 3.4.3 Leading Order Solution

The equations for the leading term can be obtained from (3.20), by setting $\epsilon = 0$. The equations read

\[
\partial_{\hat{r}} \hat{u}_{0,\hat{r}} + \frac{1}{\hat{r}} \partial_{\hat{r}} \hat{u}_{0,\hat{r}} - \text{Re} \partial_{\hat{x}} \hat{p}_0 = 0, \tag{3.28a}
\]

\[
\text{Re} \partial_{\hat{x}} \hat{p}_0 = 0, \tag{3.28b}
\]

\[
\partial_{\hat{x}} \hat{u}_{0,\hat{x}} + \partial_{\hat{r}} \hat{u}_{0,\hat{r}} + \frac{\hat{u}_{0,\hat{r}}}{\hat{r}} = 0. \tag{3.28c}
\]

From (3.28b), we conclude that $\hat{p}_0$ is only function of $\hat{x}$, and after multiplying (3.28a) by $\hat{r}$ and integrating with respect to $\hat{r}$ we get

\[
\hat{r} \partial_{\hat{r}} \hat{u}_{0,\hat{x}} = \text{Re} \partial_{\hat{x}} \hat{p}_0 \frac{\hat{r}^2}{2} + c_1(\hat{x}). \tag{3.29}
\]

By evaluating the previous expression at $\hat{r} = 0$ we find $c_1(\hat{x}) \equiv 0$, and integrating once more with respect to $\hat{r}$ we get

\[
\hat{u}_{0,\hat{x}} = \text{Re} \partial_{\hat{x}} \hat{p}_0 \frac{\hat{r}^2}{4} + c_2(\hat{x}). \tag{3.30}
\]

Finally, using the no-slip condition at the wall of the pipe, we can determine the function $c_2(\hat{x})$, and we obtain

\[
\hat{u}_{0,\hat{x}} = \frac{\text{Re} \partial_{\hat{x}} \hat{p}_0}{4} \left( \hat{r}^2 - \hat{R}_w^2(\hat{x}) \right). \tag{3.31}
\]

Combining (3.27), and (3.31) we obtain the following expression for the pressure gradient $\partial_{\hat{x}} \hat{p}_0$

\[
\partial_{\hat{x}} \hat{p}_0 = -\frac{2}{\text{Re} \hat{R}_w^4(\hat{x})}. \tag{3.32}
\]

Consequently, $\hat{u}_{0,\hat{x}}$ takes the form

\[
\hat{u}_{0,\hat{x}}(\hat{x}, \hat{r}) = \frac{1}{2\hat{R}_w^4(\hat{x})} \left( \hat{R}_w^2(\hat{x}) - \hat{r}^2 \right). \tag{3.33}
\]

Finally, from (3.28c), we can determine the radial velocity component $\hat{u}_{0,\hat{r}}$. First from (3.33) we derive

\[
\partial_{\hat{x}} \hat{u}_{0,\hat{x}} = \frac{\left( 2\hat{r}^2 - \hat{R}_w^2(\hat{x}) \right) \hat{R}_w'(\hat{x})}{\hat{R}_w^2(\hat{x})}. \tag{3.34}
\]
Substituting this expression in (3.28c), integrating w.r.t. \( \tilde{r} \) and using the no-slip condition we get

\[
\tilde{u}_{0,r} = \frac{\tilde{r} \left( \dot{R}_w(\tilde{x}) - \tilde{r}^2 \right) \dot{R}_w(\tilde{x})}{2 \dot{R}_w(\tilde{x})} = \frac{\tilde{r} \ddot{R}_w(\tilde{x})}{\dot{R}_w(\tilde{x})} \tilde{u}_{0,\tilde{r}}(\tilde{r}, \tilde{x}).
\]  

(3.35)

Summarizing, the 0th order terms of the asymptotic expansion are

\[
\begin{align*}
\tilde{u}_{0,\tilde{r}}(\tilde{x}, \tilde{r}) &= \frac{1}{2 \dot{R}_w(\tilde{x})} \left( \dot{R}_w(\tilde{x}) - \tilde{r}^2 \right), \\
\tilde{u}_{0,\tilde{x}}(\tilde{x}, \tilde{r}) &= \tilde{r} \ddot{R}_w(\tilde{x}) \tilde{u}_{0,\tilde{r}}(\tilde{r}, \tilde{x}), \\
\tilde{p}_0(\tilde{x}, \tilde{r}) &= -\frac{2}{Re} \int_0^{\tilde{x}} \frac{1}{R_w^4(\xi)} d\xi.
\end{align*}
\]

(3.36a, 3.36b, 3.36c)

This particular expression for \( \tilde{p}_0 \) considers setting a reference pressure \( \tilde{p}_0(0, 0) = 0 \). These expressions produce an approximation for the original dimensional variables \( u_x, u_r \) and \( p \), i.e.,

\[
\begin{align*}
{u}_x(x, r) &= 2U \frac{R_w^2(0)}{R_w^2(x)} \left( 1 - \frac{r^2}{R_w^2(x)} \right), \\
{u}_r(x, r) &= \frac{R_w(x)}{R_w(x)} {u}_x(r, x), \\
{p}(x) &= -\frac{16 \rho U_x^2 R_w^4(0)}{Re} \int_0^{x} \frac{1}{R_w^4(\xi)} d\xi.
\end{align*}
\]

(3.37a, 3.37b, 3.37c)

### 3.4.4 Estimation of the Friction Factor

In this section we consider two different ways of using the asymptotic solution derived above, in order to find the pressure drop. Naturally the first idea that comes in mind is to directly use expression (3.37c) and evaluate it at \( x = L \), thus finding the correspondent pressure drop. The other possibility we consider, is to use the leading terms of the asymptotic expansion (3.37) for evaluating the integrals in (3.5). The first option leads to a result which was originally obtained by Deiber et al. [25]. This expression, however, is not able to capture the dependency on the period \( L \), as we will see in our case study featuring sinusoidal pipes. The second alternative, is able to capture the dependency on the period \( L \), thus extending the region of applicability of the method. Now we proceed to obtain the two approximations.

Following the first idea, using that \( \tilde{p}_0 \) is independent of \( \tilde{r} \) and evaluating (3.37c), the
3.4 An Asymptotic Formula for the Friction Factor

The Darcy friction factor can be obtained by solving for \( f \) in (3.4), this yields

\[
f = \frac{64}{\text{Re}} \frac{R_w^4(0)}{L} \int_0^L \frac{1}{R_w^4(x)} \, dx. \tag{3.39}
\]

where \( \text{CF1} \) can be interpreted as a \textit{correction factor}, which when multiplied with the friction factor \( 64/\text{Re} \) for laminar flow in straight pipes, gives us an approximation to the friction factor of an arbitrarily shaped axially symmetric periodic pipe, described by the function \( R_w(x) \). As mentioned before, (3.39) corresponds to the result previously obtained by Deiber et al. [25]. One of the main advantages of such an estimation, is that it only requires the calculation of a one-dimensional integral, consequently obtaining a huge reduction in computation time, when compared to general CFD type methods.

In order to analyze how the approximation with \( \text{CF1} \) performs, we compare the results obtained with (3.39) to those obtained with the numerical model described in Section 3.3. For the simulations we consider sinusoidal pipes. Figure 3.7 shows a plot of the friction factor versus the Reynolds number for a sinusoidal pipe with amplitude \( a = 2 \).

**Figure 3.7:** Friction factor (solid lines) and approximation obtained with correction factor \( \text{CF1} \) (3.39)(dotted lines), as function of the Reynolds number, for a sinusoidal pipe depicted as in Figure 3.14. Parameter values are \( D = 2 \), and \( a = 1 \). The estimation obtained with \( \text{CF1} \) is the same for all the values of \( L \).

**Figure 3.8:** Friction factor (solid lines) and approximations obtained with correction factor \( \text{CF2} \) (3.46) (dotted lines), as function of the Reynolds number, for a sinusoidal pipe depicted as in Figure 3.14. Parameter values are \( D = 2 \), and \( a = 1 \). The estimation with \( \text{CF2} \) is able to capture the changes in friction factor due to changes in the period \( L \).

The total pressure loss becomes

\[
\Delta p = \frac{16\rho U_x^2 R_w^4(0)}{\text{Re}} \int_0^L \frac{1}{R_w^4(x)} \, dx. \tag{3.38}
\]
and different values for the period $L$. We can notice the deviation of the friction factor computed with CFD (solid lines), from the friction factor for straight pipes $64/Re$ (dotted line). The friction factor obtained with (3.39) does not captures the dependency on the parameter $L$, and it provides the same result regardless of the value of $L$. Nonetheless, we can observe how the formula predicts the correct value when the period of the pipe $L$ increases. Now we proceed with the second alternative.

Instead of using the asymptotic solution directly, we substitute the asymptotic solution (3.37) into the integral expression for the pressure drop (2.35) and calculate the integrals. First we derive the pressure loss due to pressure forces on the wall $\Delta p_p$. Using the expressions for the normal vector (3.2a) and the surface element (3.2b), we obtain

$$\Delta p_p := \frac{1}{|\Gamma_{in}|} \int_{\Gamma} \rho u_x dS = \frac{32\rho U_x^2 R_w(0)}{Re} \int_0^L \left( \int_0^x \frac{1}{R_w(\xi)} d\xi \right) R_w(x) R_w'(x) dx.$$  

(3.40)

Changing the order of integration we get

$$\Delta p_p = \frac{16\rho U_x^2 R_w(0)}{Re} \left[ R_w^2(L) \int_0^L \frac{1}{R_w^2(x)} dx - \int_0^L \frac{1}{R_w^2(x)} dx \right].$$  

(3.41)

In the same way, using (3.37) and (3.2a), we can obtain the pressure loss due to skin friction $\Delta p_S$. First we compute

$$\partial_r u_x = -4U_x \frac{R_w^2(0)}{R_w^4(x)},$$

$$\partial_x u_x = 4U_x \frac{R_w^2(0)}{R_w^4(x)} \left[ 2 \frac{r^2}{R_w^4(x)} - 1 \right].$$  

(3.42)

Then we can evaluate $\partial_n u_x$ at the wall $\Gamma$ and get

$$\Delta p_S := -\frac{\mu}{|\Gamma_{in}|} \int_{\Gamma} \partial_n u_x dS = 8\mu U_x \int_0^L \frac{1}{R_w^2(x)} \left[ 1 + (R_w'(x))^2 \right] dx.$$  

(3.43)

Adding the pressure loss due to forces on the wall (3.41) with the pressure loss due to skin friction (3.43), we get the following approximation for the total pressure loss

$$\Delta p = \frac{16\rho U_x^2 R_w(0)}{Re} \left[ R_w^2(L) \int_0^L \frac{1}{R_w^2(x)} dx - \int_0^L \frac{1}{R_w^2(x)} dx \right] +$$

$$+ 8\mu U_x \int_0^L \frac{1}{R_w^2(x)} \left[ 1 + (R_w'(x))^2 \right] dx.$$  

(3.44)
3.4 An Asymptotic Formula for the Friction Factor

Grouping terms and using that $\rho DU_x/\mu \text{Re} = 1$, we finally get

$$\Delta p = \frac{16\rho U_x^2 R_w(0)}{\text{Re}} \int_0^L \left( \frac{R_w'(x)}{R_w^2(x)} \right)^2 + \frac{R_w^2(0)}{R_w^4(x)} \, dx,$$

which in terms of a friction factor yields

$$f = \frac{64}{\text{Re}} \frac{R_w^2(0)}{L} \int_0^L \left( \frac{R_w'(x)}{R_w^2(x)} \right)^2 + \frac{R_w^2(0)}{R_w^4(x)} \, dx.$$

This gives us an alternative expression for approximating the friction factor, that in contrast with (3.39), is able to capture the dependency on the parameter $L$, as we can be observe in Figure 3.8. The estimations obtained with our new approximation (3.46) are displayed in dotted lines, and the results obtained with CFD in solid lines, the line corresponding to $64/\text{Re}$ is displayed for reference. The estimate (3.39) accurately predicts the friction factor up to certain Reynolds number, at which inertial effects become more important.

We again compare the results against the experimental data from [25]. In Figure 3.9 we show a plot of experimental data (blue dots), and two-dimensional numerical simulation (green solid line), obtained by Deiber et al. [25, p. 642]. The estimation obtained with (3.46) is displayed with a red dashed line. The two-dimensional numerical simulation by Deiber et al., was obtained by means of an iterative technique in which the shape is slowly altered from a reference configuration. Their experiments, were conducted in a sinusoidal pipe, which in terms of our parameters correspond to $a = 0.8571$ and $L = 8.9714$, see Figure 3.14. The estimation obtained with (3.46) matches both plots up to a Reynolds number of approximately $\text{Re} \approx 200$. For larger Reynolds numbers, the estimation starts to deviate from the experimental data, while the two-dimensional numerical solution follows to some extend, till at $\text{Re} \approx 750$, a change towards turbulence can be observed from the experimental data, and nor the estimate, nor the numerical solution, are expected to correctly describe the flow.

One of the most important advantages of (3.46) is that it is able to follow the effects due to changes in the period $L$, while the computational cost are still those of numerical integration in one dimension. For instance a typical computation time for (3.46), with an adaptive quadrature rule, with a tolerance of $\text{tol} = 10^{-6}$ in MatLab, using a computer with an Intel Quad Core 2.4[GHz] processor and 4[GB] of RAM, ranges from $6.01 \times 10^{-3}$, to $7.83 \times 10^{-3}$ seconds. In contrast, the computation times of the numerical model presented in Section 3.3, range from 20, to 51 seconds, for a fine mesh (1039 mesh points), and from 11, to 36 seconds, for a coarse mesh (212 mesh points), just to mention some examples. The parameters of the geometry for this computations were varied from $a = 0$ to $a = 2$, $L = 1.5$ to $L = 80$, and $\text{Re} \approx 10$ to $\text{Re} \approx 1200$. 
3.4.5 Experimental and Numerical Validation

In order to systematically investigate the accuracy and range of applicability of our approximation (3.46) to the friction factor, we consider a case study featuring sinusoidal pipes, in which the geometrical parameters were varied from 0 to 2 for the amplitude of the pipe $a$, and from 1.5 to 80 for the period of the pipe $L$. In this case study, the geometry had been previously rescaled in such a way that $R_w(0) = 1$. For each of these geometries, we computed the friction factor using the numerical model, and using the asymptotic formula (3.46), which allowed us to compute the relative error $Err$ as

$$Err := \frac{|f - \tilde{f}|}{|f|},$$

(3.47)

where $f$ is the friction factor obtained from the numerical model, and $\tilde{f}$ is the estimated friction factor obtained with (3.46).

The results from these tests are shown in Figure 3.10. The regions in the parameter space, were the method delivers approximations with relative errors $Err = 1\%$, $Err = 10\%$, and $Err = 20\%$ are presented as iso-surfaces. The zones below each of the iso-surfaces, constitute a region where our approximation yields a relative error smaller than the corresponding error of the iso-surface. For instance, if the period of the pipe is $L = 80$, and the Reynolds number $Re = 50$, our approximation yield and error smaller
than $\text{Err} = 1\%$, for any amplitude $0 \leq a \leq 1$.

In order to give a more clear impression of the regions of accuracy of the method, we show cross sections of the error for some fixed values of the amplitude $a$, as function of $\text{Re}$ and $L$. The results are displayed in terms of contour lines of the error. Figure 3.11 shows the results for the case $a = 0.2$. Some remarkable property, is the fact that the maximum error in the whole region is only 8%. Of course this accuracy can not be attained for all parameter values. When one increases the size of the amplitude, the accuracy of the method decreases, for instance when $a = 0.5143$, Figure 3.12, there are still some regions where the accuracy is of the order of 5%, but in other regions the error increases up to 25%. For the case $a = 1$, Figure 3.13, the region of 5% accuracy is reduced, and some zones with error of up to 30% appear.

We note that for large values of $\text{Re}$, and $a/L$, (i.e. in the lower right corners in Figures 3.13, 3.11 and 3.12), the flow could already be in the turbulence region. In such kind of situation, the assumption of steady flow is no longer valid and consequently the estimated friction factors are unlikely to be accurate.

### 3.4.6 Applicability of the Method

Based on asymptotic solutions obtained via the method of slow variations, and on an integral expression for the friction factor, we derived an approximate analytical formula for the friction factor in axially symmetric pipes. The formula is able to capture the
dependency of the friction factor on the period of the pipe, and it is computationally efficient, because it only requires numerical integration in one dimension. A typical computation time of 11 seconds for two-dimensional numerical simulations, can be reduced to just $6 \times 10^{-3}$ seconds, when using this approximation.

From the validation of sinusoidal pipes presented in this section, we can conclude that the formula yields an error smaller than 10%, for amplitude values $a \leq 0.2$, provided that $L \geq 1.5$, and $Re \leq 750$. For larger amplitudes, we additionally require, either a small Reynolds number Re, or a large value of $L$, in order to keep the error below 10% [102]. The maximum error, considering the range of parameters investigated here, is about 25% and 30%, for amplitudes $a = 0.5143$ and $a = 1$, respectively.
3.5 Wall-Shape Design

In this section, we study the effect of wall-shape on the flow, and consider the possibility of finding a wall-shape that maximizes the flow rate for a given pressure gradient. We discuss the effect that each of the geometrical parameters has on the flow; in parallel, we address the influence of the Reynolds number. We present the results for two different geometrical configurations, the cavity configuration, and the slowly varying geometry. The role of the parameters in each case is discussed in detail. In both cases it turns out possible to increase the flow rate. The situation is specially promising in the case of the slowly varying geometry, for which an improvement of up to 120\% was observed. We also show that for large pressure gradients, an asymmetric wall-shape performs better than the symmetric counterpart. We will also study the dependency of the optimal ratio of expansion and contraction, on the pressure gradient.

As stated before, the main goal of this section is to characterize the effect of wall-shape on the flow and to study the possibility of designing a wall-shape that maximizes the flow rate for a given applied pressure gradient. In order to study the effects of wall-shape, we will consider a piecewise sinusoidal pipe depicted as in Figure 3.14 [91]. The shape consists of one expansion region of length $E$, and a contraction region of length $C$. The expansion part can be considered as the first half of a sinusoidal pipe with period $2E$, and likewise, the contraction part can be considered as the second half of a sinusoidal pipe with period $2C$. These parts are matched at $x = E$, in such a way that both $R_w$ and $R'_w$ are continuous. The geometry considered here, has already been scaled by the minimum radius of the pipe, i.e. $R_w(0) = 1$, and the period of the pipe can be computed as $L = E + C$.

![Figure 3.14: Periodic section of a pipe with center line along the x-axis, $a$ represents the amplitude of the corrugation, $E$ the length of the expansion region, $C$ the length of the contraction region. The period $L$ of the pipe, satisfies $L = E + C$.](image)

To classify the geometry, we will use the amplitude of the pipe $a$, the period of the pipe $L$, and the quotient $L_e$ of the length of the expansion region over the length of the
Isothermal Laminar Flow

period, which is defined as

\[ L_e := \frac{E}{L} \]  

(3.48)

These coefficients completely define the geometry, and also allow us to distinguish between two essentially different, geometric configurations. The categories we will refer to are the cavity configuration, i.e., \( L \ll 1 \), and the slowly varying configuration\( \)slowly varying configuration, which appears when \( L \gg 1 \). As we will show later, these two configurations exhibit substantial differences in the flow behavior. For example, in the slowly varying case, the parameter \( L_e \) plays a more important role on the final flow rate than in the cavity case. This classification, basically allows us to identify the cases, in which wall-shape design can deliver a significant increase on the flow rate.

In fact, from the experimental results presented in Figure 3.5, we can already observe that the friction factor of a wavy pipe with radius 1, amplitude \( a = 0.8571 \), and period \( L = 8.9714 \), is smaller than the friction factor of a straight pipe with radius 1. The reduction in friction is about 50% and 30%, at \( Re \approx 81.33 \) and \( Re = 612.7 \), respectively. This suggests that the friction factor can be reduced. Therefore, we want to determine in which situations it is possible to reduce the friction factor (or increase the flow rate \( Q \) for given pressure gradient), and more specifically, to find the optimal set of design parameters \( a, L, \) and \( L_e \), i.e., the ones that maximize the flow rate \( Q \), for a given pressure gradient \( \beta \).

### 3.5.1 The Cavity Configuration

Now we will analyze the effect of the parameters \( a \) and \( L_e \), in the case of the cavity type configuration, i.e., \( L \ll 1 \). In our particular case, we consider a geometry which after scaling, corresponds to a sinusoidal pipe with parameters \( L = 0.2666 \), and \( L_e = 0.5 \). The wall-shape of this pipe has an amplitude of \( a = 0.1053 \), but instead of considering only this fixed value, we will vary this parameter and describe its influence.

In Figure 3.15, we present a plot of the flow rate \( Q \), versus the mean pressure gradient \( \beta \), for different values of \( a \). All flow rates have been scaled by the reference flow rate \( Q_{sp} \), which corresponds to the flow rate in a straight pipe with radius 1, see (3.8). From this figure, we can observe that for \( a = L/5 \), the flow rate is about 5.5% larger than in the reference straight pipe. This increment in the flow rate continues for increasing values of \( a \), but the extra increment is every time smaller. For instance, when we set \( a = L \), the increment in flow rate is about 7.8%, whereas as if we set \( a = 2L \) then the total increment in flow rate is about 8.3%, i.e., after increasing the amplitude twice, the extra increment we obtain is only 0.5%.

This numerical evidence, leads us to conclude that the flow rate always increases with the amplitude, and that the increase is every time smaller. Of course this statement is
3.5 Wall-Shape Design

Figure 3.15: Flow rate $Q$ as function of the applied pressure gradient $\beta$, for sinusoidal pipes with $L = 0.26$, $L_e = 0.5$, and several values of $a$. All values have been scaled with the reference flow rate $Q_{sp}$ of a straight pipe with radius 1 (see (3.8)).

Figure 3.16: Pressure fluctuations $\tilde{p}$, and streamlines for a sinusoidal pipe with radius at inlet $R_w(0) = 1$, amplitude period $L = 0.26$, $L_e = 0.5$ and various amplitudes $a$. All pictures are in the same scale, and in order to display the vortices clearly, the left limits of all pictures start from $R = 0.9$. The flow direction is upwards.
based on the assumption that the flow remains laminar. This condition is very important, because we cannot expect this statement to hold for any arbitrary large Reynolds number. In case the flow exceeds the so-called critical Reynolds number, the flow is expected to become turbulent, in which case, the friction would be larger and the flow rate would reduce. In fact, there is strong evidence that increasing the amplitude \( a \), might cause a reduction of the value of the critical Reynolds number \( \text{Re}_{cr} \). We can get some idea of why this might occur by looking at the changes in the flow pattern for different values of \( a \). In Figure 3.16, we can observe the streamlines for a pipe with period \( L = 0.26 \), and different amplitudes. From this picture we can see how for the same Reynolds number \( \text{Re} \in [0.84, 0.85] \), the complexity of the flow increases with increasing \( a \). For instance, when \( a = L/2 \) (Figure 3.16(a), we can observe a small vortex, when we increase the amplitude to \( a = L \) (Figure 3.16(b)), and \( a = 1.8L \) (Figure 3.16(c)), we can observe two and three vortices respectively.

Another interesting observation is that when \( a \) increases, the Reynolds number at which a vortex appears decreases. For example, at \( a = L/5 \) and \( \text{Re} = 0.84 \), the flow exhibits no vortex, whereas for \( a = L/2 \) and the same Reynolds number, in Figure 3.16(a) we can observe a small vortex. This vortex increases in size with \( \text{Re} \), as we can observe in Figure 3.16(d). For the cases \( a = L, 1.8L \), even when \( \text{Re} \approx 0.85 \) the vortices completely fill the protrusion of the pipe. In these cases, the effect of increasing \( \text{Re} \) is that the vortex in "contact" with the core looses symmetry. A similar situation can be observed in the pressure field. For the low Reynolds number \( \text{Re} = 0.85 \), the pressure field is almost antisymmetric with respect to \( x = L/2 \). When the Reynolds number increases this property is lost. For example, at \( \text{Re} \in [265, 268] \), we can clearly see in Figures 3.16(b), 3.16(d), that the red region covers a larger region than the blue region.

Now we study the influence of the parameter \( L_e \). In Figure 3.17(a) we have plotted the flow rate, scaled by \( Q_{sp} \), for a fixed amplitude \( a = L/5 \) and various values for \( L_e \). As we can see, \( L_e \) does not play a very important role in this configuration, and changing \( L_e \) from the value 0.5 to another value, appears to be detrimental. For instance, setting \( L_e = 0.1 \), reduces our original increase in flow rate from 5.5% to 5%, and a similar effect can be observed when we set \( L_e = 0.76 \). But this is not always true, in fact, for \( \beta = 10^3 \), setting \( L_e = 0.53 \) delivers a small improvement. In order to see this clearly, for each value of \( \beta \), we have normalized the flow rates by the maximum flow rate, and we have plotted this normalized flow rate versus the parameter \( L_e \), for different values of \( \beta \) in Figure 3.17(b). For \( \beta = 1 \), the flow rate is symmetric with respect to \( L_e = 0.5 \), and at this point the maximum flow rate is attained. When \( \beta = 10^3 \), the flow rate shows an asymmetric shape, and the optimal value shifts to \( L_e = 0.53 \), which is indicated by an arrow. Unfortunately, as we can see from Figure 3.17(a), the additional increment in flow rate is very small.

The flow rate as function of \( L_e \) becomes more symmetric for large values of \( a \). In Figure 3.17(c), we can see the normalized flow rates for \( a = L \). From this picture, we can
3.5 Wall-Shape Design

Figure 3.17: Flow rates (scaled with $Q_{sp}$) versus the pressure gradient $\beta$ for different values of $L_e$. Figures 3.17(b),3.17(c), show the flow rates $Q$, as function of $L_e$. Here the flow rates were scaled with the maximum flow rate $Q_{max}$ for each value of $\beta$. The parameter $L$ in both figures is $L = 0.26$. 
Figure 3.18: Flow rate $Q$ as function of the applied pressure gradient $\beta$, for sinusoidal pipes with $L = 5$, $L_e = 0.5$, and several values of $a$. All values have been scaled with the reference flow rate $Q_{sp}$ of a straight pipe with radius 1, see (3.8). For comparison, the configuration with largest flow rate from Figure 3.15 (blue triangles) has been added.

Clearly see that even for $\beta = 10^3$, the curve for the flow rate remains symmetric and the optimum is attained at $L_e = 0.5$. For the cavity configuration, the parameter $L_e$, can only give some improvement when $a$ is small, but this improvement is so small, that in practice it is simply better to set $L_e = 0.5$. In the next section, we will see that the situation is different for the slowly varying configuration.

3.5.2 The Slowly Varying Configuration

Now we study the effect of the parameters $a$ and $L_e$ for the slowly varying configuration. To this extend, we consider a family of sinusoidal pipes with parameter $L = 5$. As in the case of the cavity configuration, we study first the effect of varying the amplitude $a$. In Figure 3.18, we can observe the obtained flow rate $Q$, scaled by $Q_{sp}$, versus the pressure gradient $\beta$, for different values of $a$. For comparison, we have added the curve with highest flow rate from the cavity configuration, i.e., the plot corresponding to the parameters $L = 0.26$, and $a = 3.2L = 0.83$ from Figure 3.15. From this figure, we immediately realize that $a$ has much more influence than in the case of the cavity configuration.

The increment in flow rate for the cavity geometry with parameters $L = 0.26$, and $a = 0.83$, delivers an increment of 8.6%, relative to a straight pipe. This increment is not very substantial when compared to the results obtained with the slowly varying geometries.
3.5 Wall-Shape Design

For instance, when $L = 5$, and $a = 0.2$, the increment in flow rate is about 39%, for small values of $\beta$, and about 24% for the largest value of this parameter, namely $\beta = 10^{3.1}$ (Re = 985). Even though the amplitude in this slowly varying configuration is about one quarter of the amplitude in the cavity configuration, the increment in flow rate is between 2 and 5 times higher. This effect becomes much more prominent for larger values of $a$, going from a 73% increment for $a = 0.4$, to a 120% improvement for $a = 0.8$, for small pressure gradients. For the largest applied pressure gradient of $\beta = 10^{3.5}$, the increments in flow rate are 31%, 33%, and 35%, for the geometries with $a = 0.4$, $a = 0.6$, and $a = 0.8$, respectively.

If we concentrate only on the slowly varying geometries, again, as in the cavity configuration case, we can conclude that the flow rate always increases when the amplitude $a$ increases. This increment shows the same behavior, as in the cavity configuration case, and the extra increments are smaller each time. This effect becomes stronger for large values of $\beta$, as we can see from Figure 3.18, this happens because as $\beta$ becomes large, inertia effects become more important, and vortices are expected to appear. As reference, we have added small arrows in Figure 3.18, indicating the smallest value of $\beta$ in which a vortex appears.

These simulations also suggest that the flow is more likely to remain laminar than in the cavity case. To see this, let us first recall that in the cavity case, for $a = L/2 = 0.13$, the Reynolds number at which a vortex appears is Re = 0.84 (see Figure 3.16). This value is much smaller, than the Re = 372 for the geometry with $L = 5$, and $a = 0.4$. This basically means that the assumption of laminar flow holds in a larger range of Re.

Now we turn our attention to study the influence of the parameter $L_e$. Figure 3.19 shows the scaled flow rate versus the pressure gradient $\beta$ for $a = 0.8$, and three different values of $L_e$. As we can observe from this figure, the role of $L_e$ is a little more important than in the cavity configuration. Even though the changes in flow rate are not very large, the parameter $L_e$ plays an interesting role. For instance, for $\beta = 10^{2.5}$ (Figure 3.19(b)), the pipe with $L_e = 0.65$ delivers an increment in flow rate of 69%, i.e., 1% more than the pipe with $L_e = 0.5$. On the other hand, for $\beta = 10^{3.5}$, now the pipe with $L_e = 0.75$, is the best pipe among these three, with an improvement of 36.7% in flow rate, compared to the 35.5% of the pipe with $L_e = 0.5$, and the 36% of the one with $L_e = 0.65$.

The previous observation lets us see that the optimal value of $L_e$ depends on the mean pressure gradient $\beta$. This dependency can be seen more clearly in Figure 3.20. In this figure we have plotted the flow rates as functions of $L_e$. For convenience, all values have been scaled by the maximum flow rate, for each value of $\beta$. The maximum of each curve appears indicated by an arrow, and from this figure, it is clear that the optimal $L_e$ shifts to the right when $\beta$ increases, for instance, for $\beta = 10^2$, the largest flow rate is attained around $L_e = 0.6$, and for $\beta = 10^{3.5}$, around $L_e = 0.8$. In the case of small pressure gradients, the optimal value of $L_e$ approaches 0.5. It is also worth mentioning that in
 Isothermal Laminar Flow

Figure 3.19: Flow rates (scaled with $Q_{sp}$) versus the pressure gradient $\beta$ for different values of $L_e$. Figures 3.19(b), and 3.19(c), show the plot zoomed around $\beta = 10^{2.5}$, and $\beta = 10^{3.5}$ respectively. The parameter $L$ was set to 5.

all cases, including the cavity configuration, setting $L_e$ below 0.5 is always detrimental, and therefore this should be avoided.

3.5.3 Remarks on Wall-Shape Design

In this section we have studied the effect of wall-shape on the flow. With our numerical model, we could tackle the problem of finding a wall-shape of a periodic section of a pipe, for improving the flow rate for a prescribed pressure gradient. From the two parameters characterizing the geometry, the amplitude $a$ proves to be the one that contributes the most for increasing the flow rate. The potential improvement is specially promising when the period is larger than the radius of the pipe. In our case study, we observed an increase in flow rate of up to 120%, for small Reynolds numbers, and
of 35%, for large Reynolds numbers. When the period is smaller than the radius, the improvement was about 8%.

The ratio between the expansion and contraction regions, turns out to be irrelevant for the cavity configuration, and in this case, it is better to concentrate on finding a convenient value for the amplitude. In practice, one would need to balance the benefit of increasing the flow rate, with the extra costs of increasing the amplitude \(a\). In the slowly varying case, the parameter \(L_c\) shows a very interesting behavior. For large values of the pressure gradient, it is possible to obtain some extra improvement by modifying the wall-shape from a symmetric shape \((L_c = 0.5)\), to a geometry where the expansion region is longer than the contraction region \((L_c > 0.5)\).

Overall, the possibility of improving the flow rate of laminar flow in corrugated pipes by modifying the wall-shape is very promising. In general, we can say that in order to benefit the most from wall-shape design, it is recommendable to attempt to increase the period of the pipe. Of course, in a practical situation, the benefits of obtaining a larger flow rate should be balanced with other possible effects and technical restrictions, such as mechanical strength, among others.
Chapter 4

Non-Isothermal Laminar Flow

In the present chapter we tackle the problem of non-isothermal flow, in other words, we are concerned, not only with the flow field, but also with the transfer of energy in the form of heat. First, we discuss the problem of laminar forced convection with negligible buoyancy contribution. This leads to a one-way coupling between the flow equations and the energy equation. Practically this means that one can first solve the flow equations and then impose the velocity field as a forced convection term in the energy equation. We discuss how to handle Neumann type boundary conditions, i.e., problems with prescribed heat fluxes at the boundaries. We show that the computational domain can be reduced to just one period thereby reducing the computational costs of a numerical model.

Next, we discuss the modeling of natural convecting flow in a vertically oriented geometry. The need of taking into account buoyancy forces introduces a two-way coupling between the flow equations and the energy equation. We consider a real-life application featuring a thermosyphon loop and present a numerical model for simulating this kind of flow. We show that it is possible to achieve a better performance of the thermosyphon loop by tuning the wall-shape. However, the computational cost for this kind of simulation increases significantly and in this case it is no longer possible to reduce the computational domain by using a periodicity type argument unlike in the case of laminar forced convection. This problem requires other techniques for reducing the computational costs. In Chapter 5, we present a more efficient alternative based on the method of homogenization.
4.1 Convective Heat Transfer

Convective heat transfer refers to the combination of conduction and advection. Convection is typically classified into forced convection and natural convection. The term forced convection (see Section 2.4.1) refers to the situation in which the movement of the fluid is caused by an external source such as a fan or a pump, while the term natural convection (see Section 2.4.2) refers to the situation in which the flow is a result of buoyancy forces. In any case, the motion of fluid can contribute significantly to the transport of heat and that is why knowing the velocity field is crucial in convection problems. This means that we need to consider both, the fluid flow equations and the energy equation, to address convective heat transfer.

Depending on whether one is dealing with a forced or with a natural convection problem, different simplifications and approaches can be applied. In Section 2.4, we discussed how each case leads to different systems of equations. The main difference between the system of equations describing forced convection (eq. (2.37)) and the one describing natural convection (eq. (2.41)) is the appearance of the buoyancy term $-g/\rho T^*$ in the momentum equations of the natural convection system (2.41b).

The buoyancy term is very significant, because it implies a two-way coupling between the flow and temperature equations. In practice this means that we have to solve all the equations simultaneously in the case of natural convection. On the other hand, in the case of forced convection, there is only a one-way coupling via the advection term in the temperature equation. In practical terms this means that we can solve the flow equations independently from the temperature equation. Once the velocity field is known, we can introduce the advection term in the temperature equation and solve for the temperature field. In this sense the problem of forced convection is simpler than the one of natural convection. We will now discuss the problem of forced convection and in a later section, we will proceed with the case of natural convection.

4.2 Forced Convection

There are different questions which arise in connection with convective heat transfer problems. One can for instance be interested in knowing the losses of mechanical energy (pressure drop) in the flow or in finding the thermal resistance to heat transfer in the direction normal to a given flow [75].

In other problems, the wall heat flux is given and one is interested in knowing the temperature variation along the wall. Such a problem arises particularly in the application of cooling of electrical and nuclear systems [6]. In these type of applications overheat-
4.2 Forced Convection

![Diagram of a periodic pipe with center line along the x-axis and period L. The flow is driven by the mean axial pressure gradient $\beta$ and $q_w$ represents the fixed wall heat flux normal to the surface. We solve the problem in a single period of the pipe, denoted by $\Omega$, the wall of the pipe of this period is denoted by $\Gamma$. The inlet and outlet of this periodic section are denoted by $\Gamma_{in}$ (cross section at $x = 0$) and $\Gamma_{out}$ (cross section at $x = L$), respectively.]

Figure 4.1: Periodic pipe with center line along the x-axis and period L. The flow is driven by the mean axial pressure gradient $\beta$ and $q_w$ represents the fixed wall heat flux normal to the surface. We solve the problem in a single period of the pipe, denoted by $\Omega$, the wall of the pipe of this period is denoted by $\Gamma$. The inlet and outlet of this periodic section are denoted by $\Gamma_{in}$ (cross section at $x = 0$) and $\Gamma_{out}$ (cross section at $x = L$), respectively.

...ing, burnout or meltdown are crucial. In order to prevent these undesirable situations, the devices should be designed in such a way that the temperature is always below a certain safety value. A similar situation arises when dealing with the transfer of LNG through pipelines. In this application, one could be interested in keeping the temperature of the LNG below the saturation limit in order to prevent phase transition. The difference is that now one is interested in knowing the temperature field everywhere in the domain rather than just at the wall of the pipe. However, even when one is only interested in the temperature at the wall, we must first determine the flow and temperature fields everywhere in the domain in order to find the temperature at the wall.

4.2.1 Uniform Heat Flux Problem

We have seen that the problem of fixed heat flux is meaningful for various applications. We now discuss how this problem can be addressed. We consider forced flow along a corrugated pipe depicted as in Figure 4.1. The flow is driven by a prescribed mean axial pressure gradient $\beta$. Heat is transferred to the flow stream through a fixed constant wall heat flux $q_w [\text{W m}^{-2}]$ at $\Gamma$.

We need to solve the system of equations describing forced convection in order to solve this problem. For convenience, we recall the system of equations describing forced con-
Non-Isothermal Laminar Flow

In dimensional form and in steady state it reads

\[ \nabla \cdot \mathbf{u} = 0, \]  
(4.1a)
\[ \rho_{\text{ref}} (\mathbf{u} \cdot \nabla \mathbf{u}) = -\nabla p + \mu \nabla^2 \mathbf{u}, \]  
(4.1b)
\[ \rho_{\text{ref}} C_p \mathbf{u} \cdot \nabla T = \kappa \nabla^2 T, \]  
(4.1c)

where \( \mathbf{u}, p \) and \( T \) are the velocity, pressure and temperature, respectively.

The previous system of equations should be provided with appropriate boundary conditions. The boundary condition describing the constant wall heat flux reads

\[ -n \cdot \mathbf{q} = -n \cdot \nabla (-\kappa T) = q_w, \text{ at } \Gamma, \]  
(4.2)

where \( q \) stands for the heat flux and \( \Gamma \) for the wall of the pipe. For the flow equations, we consider the no-slip condition

\[ u = 0, \text{ at } \Gamma. \]  
(4.3)

4.3 Solution with Periodicity Decomposition

In addition one needs to prescribe boundary conditions at the inlet and outlet of the domain. In our case, however, we are interested in the behavior when the flow is fully developed. For this case it is possible to solve the problem by considering just one single period of the pipe.

4.3.1 Flow Equations

As we mentioned before, the system of equations (4.1) has the advantage that the flow equations do not involve the temperature and hence we can first solve the flow equations and find the velocity and pressure fields. This property entitles us to use the same approaches which we presented in Chapter 3.

In Section 3.3; we showed that for fully developed flow, it is possible to impose periodic boundary conditions for the velocity. Even though the pressure is not periodic, it was possible to decompose the pressure into a linear and a periodic component in the following way

\[ p (r, x) = \bar{p} (r, x) - \beta x, \]  
(4.4)

where \( \beta \) stands for the average pressure gradient along the axial direction. By inserting
the previous decomposition into the Navier-Stokes equations, we arrived at a system of equations for which we could impose periodic boundary conditions for both the velocity and the pressure fields (see (3.12) and (3.13)). The system featured the average pressure gradient via the volume force term $\beta/\rho$ in the axial momentum equation.

### 4.3.2 Temperature Equation

We are left with the problem of solving the temperature equation (4.1c). Just as with the pressure, the temperature field along the corrugated pipe is not periodic and it changes continuously along the channel. However, it is still possible to apply a periodicity decomposition for the temperature field as well, the argument is the following.

Since the flow is periodic and the wall heat flux is constant along the pipeline, then, the heat transferred to the fluid for each periodic section should be constant as well. In the case of a *straight pipe*, this means that the temperature grows (or decreases in the case of cooling) linearly along the axial direction, i.e., $\partial_x T = \gamma$, with $\gamma$ constant [75].

In the case of a *corrugated pipe*, the situation is very similar. The main difference is that now, the temperature can have extra oscillations due to the wall-shape. In Figure 4.2, we show the expected behavior of the temperature field. Figure 4.2(a) shows the temperature profiles in the radial direction for $x = 0$, $x = L$ and $x = 2L$. The temperature difference between each subsequent curve is constant. Figure 4.2(b) shows the temperature along the axial direction along the line $r = r_0$. The temperature deviates from a straight line due to the influence of the corrugations, but on average it behaves linearly.
In mathematical terms, the previous discussion is formalized by decomposing the temperature field $T$ as

$$T(r, x) = \gamma x + \bar{T}(r, x), \quad (4.5)$$

where $\gamma [K \cdot m^{-1}]$ is the average axial temperature gradient and $\bar{T} [K]$ represents the temperature fluctuations due to the corrugations. In an analogous way as for the pressure, the previous decomposition allows us to reformulate the temperature equation in terms of the fluctuating temperature $\bar{T}$. The main advantage is that $\bar{T}$ is periodic and hence, we can reduce the computational domain to just one period.

In order to obtain the temperature equation in terms of $\bar{T}$, we have to substitute (4.5) into the temperature equation. From (4.5) we first obtain $\nabla T = \gamma e_x + \nabla \bar{T}$, in turn, from this we find that $\nabla^2 T = \nabla^2 \bar{T}$. Substituting these expression into (4.1c), we arrive at

$$\rho_{ref} C_P u \cdot \nabla \bar{T} = \kappa \nabla^2 \bar{T} - \rho_{ref} C_P \gamma u_x, \quad (4.6)$$

where $u_x$ is the axial velocity component and $-\rho_{ref} C_P \gamma u_x$ can be introduced into the equations as a volume heat source (see Section 2.2). The temperature equation is provided with the following boundary conditions

Constant wall heat flux: $- n \cdot \nabla (-\kappa T) = q_w$, at $\Gamma$, \quad (4.7)

Periodicity: $\bar{T}(r, x = 0) = \bar{T}(r, x = L)$, \quad (4.8)

where $n$ denotes the outer unit normal vector to the surface $\Gamma$ and $L$ is the period of the pipe.

### 4.3.3 Evaluation of Temperature Gradient

The only thing left in order to be able to solve the problem of forced convection with constant wall heat flux is to determine the average axial temperature gradient $\gamma$. This can be done by formulating an energy balance $[114, 120]$. In our case, we do this by integrating the temperature equation (4.1c) over a periodic section $\Omega$ of the pipe.

First we note that since the flow is incompressible, i.e., $\nabla \cdot u = 0$, then we can rewrite the advection term in divergence form, namely $\rho_{ref} C_P u \cdot \nabla T = \nabla \cdot (\rho_{ref} C_P T u)$. Furthermore, by rewriting $\nabla^2 T = \nabla \cdot \nabla T$, integrating over the domain $\Omega$ and applying the
The third integral on the left hand side becomes zero due to the no-slip condition (4.3). The first and second terms on the right hand side cancel because of periodicity. The first and second term on the left hand side can be simplified by substituting \( T = \gamma x + \tilde{T} \), the terms containing \( \tilde{T} \) cancel due to periodicity and the previous equation takes the form

\[
\rho_{ref} C_P \int_{\Gamma_{out}} \gamma L u_x dA = \int_{\Gamma} q_w dS.
\]  

This equations can be written as

\[
C_P \gamma \dot{m} = |\Gamma| q_w, \quad \text{with} \quad \dot{m} := \rho_{ref} \int_{\Gamma_{in}} u_x dA,
\]  

where \( \dot{m} \) is the mass flow rate, and \( |\Gamma| \) stands for the surface area of the shell. From (4.11) we obtain the following expression for the average axial temperature gradient

\[
\gamma = \frac{|\Gamma| q_w}{C_P \dot{m}}
\]  

This provides us with all the elements for solving the problem of forced convection with constant heat flux. The only difference with the case of isothermal laminar flow, is that in addition to the flow equations, we have to solve the periodic boundary value problem for the temperature (4.6), where \( \gamma \) is given by (4.12).

Once this periodic decomposition has been applied, the computational domain is reduced to just one single period. Solving this problem numerically represents no further challenge, when compared to the isothermal laminar flow case discussed in Chapter 3. Therefore we now turn our attention to the more challenging case of natural convection.

\section*{4.4 Natural Convection}

In this section, we discuss the modeling of a laminar thermosyphon loop with single phase. We consider a case in which the thermosyphon return line consists of a corrugated pipe. The model takes the buoyancy forces driving the flow into account via the Boussinesq approximation. Such a model can be used to predict natural convecting flows in many kind of situations, however, we focus our attention on an application
related to the storage and delivery of cryogens, a common example is LNG (Liquefied Natural Gas). We focus our attention on the effects of wall-shape on the flow and on the temperature increase inside the thermosyphon.

To this extend we determine the dependency of the flow rate and the increase in temperature, on the geometrical characteristics of the loop. The geometry considered is a set of axially symmetric corrugated pipes described by a group of parameters; namely the pipe inner radius, the period of the corrugation, the amplitude of the corrugation, and the ratio between the expansion and contraction regions of a period of the pipe. The governing equations are solved using the Finite Element Method. We use an adaptive mesh refinement technique in order to be able to capture the effects of the wall-shape. We characterize the effects of the amplitude and of the ratio of expansion and contraction regions. In particular we show that for a given fixed amplitude it is possible to find an optimal ratio of the lengths of expansion and contraction, that minimizes the maximum increase in temperature within the thermosyphon. The results show that by adequately choosing the design parameters, the performance of the thermosyphon loop can be improved.

### 4.4.1 Natural Convection Loops

Thermosyphon loops, also known as natural convection loops, are commonly employed devices in many kind of applications, ranging from solar energy utilization [70, 108] to industrial applications in nuclear reactor cooling [133]. One of the main advantages of a thermosyphon, is that the flow within the loop is created by the buoyancy forces generated by the density gradients induced by the temperature differences in the heating and cooling sections of the loop, and therefore, it does not require a pump or other device to maintain the flow.

In the cryogenic industry, thermosyphons are commonly used and since liquid cryogens are stored at temperatures far below the environmental temperature, the cryogen will boil and produce inevitable boil-off gas caused by the conductive heat loads at the walls. The boil-off gas might create a bubble, and this is undesirable close to a pump, because in such a situation, the pump will not function properly or it might even not work at all. Adding a thermosyphon return line, can be used as a mechanism in order to prevent this kind of situation from happening.

Figure 4.3 shows an example of a storage facility in which a thermosyphon return line has been added to the system, dark blue and white denote “cold” and “hot” regions respectively. When such a system is in operation, the cryogen flows out of the storage vessel towards the pump vessel, where the liquid cryogen is pumped to its final destination, a truck for example. When the facility is not in operation, the thermosyphon loop works as follows, cold liquid comes out from the bottom of the tank, and as it
4.4 Natural Convection

flows along the pipe line towards the pump vessel, the liquid absorbs heat form the environment. The temperature differences induce buoyancy forces which drive the fluid upwards along the thermosyphon return line until the fluid returns to the main vessel where the fluid cools down again and any boil-off gas is then allowed to escape from the tank. This simple mechanism serves the purpose of taking the heat from the interconnecting lines to the main storage vessel, and therefore prevents undesirable boil-off gas near by the pump vessel, which would not allow to restart the pump properly to go back to operation [99].

In brief, a thermosyphon is simply a pipe in which the flow is driven by natural convection. In our particular case we will consider the operation of such a thermosyphon loop when the thermosyphon return line consists of a corrugated pipe. Due to their flexibility, corrugated pipes provide very convenient installation and maintenance possibilities and are therefore convenient from an operational point of view. For example, in order to replace or give maintenance to the delivery pump, one can simply close the valve of the pipe, detach it from the pump, and carry out maintenance without further complications. This kind of features make corrugated pipes very appealing, however, the effects of the wall-shape need to be addressed. We will concentrate on characterizing the influence of wall-shape on the performance of the thermosyphon and we will show that the shape can be optimized for reducing the maximum temperature inside the thermosyphon.

Figure 4.3: Diagram of the LNG cryogenic storage tank featuring a thermosyphon. The return line consists of a pipe (possibly corrugated) which center line is aligned with the direction of gravity.
4.4.2 Material Properties

As working fluid for the storage facility we will consider liquefied natural gas (LNG). LNG is natural gas that has been converted temporarily to liquid form for ease of storage and transport. As a liquid, natural gas occupies only 1/600th the volume of its gaseous state, and therefore it is stored more effectively in a limited space and is more readily transported [81]. LNG is simply natural gas that has been cooled to its liquid state at atmospheric or higher pressures. The typical temperature at which LNG is stored is 106.95[K]. Depending on the application, the storage pressure might vary from 50[kPa] to 1700[kPa] [42]. In applications related to LNG fuel, a typical storage pressure is 900[kPa], we will focus our attention on this application.

In a thermosyphon the buoyancy forces are responsible for driving the flow, and these are a result of the density variations with temperature. In our particular case we are concerned with the density variations when the pressure is close to the storage pressure, i.e., 900[kPa]. The composition of LNG is predominantly methane, and therefore for practical purposes, in this study we regard the properties of methane as those of LNG. The dependency of density on temperature for methane at a constant pressure of 900[kPa], can be observed in Figure 4.4 [58]. As we can see, the density varies almost linearly for temperatures below the saturation temperature 146[K]. In order to avoid undesirable boil-off gas, it is important that the temperature of the LNG should be below saturation at all points along the pipe line.

In order to describe the density variations with temperature, we will use the Boussinesq approximation which we discussed in Section 2.3. The Boussinesq approximation describes density variations via a linearization around the reference temperature \( T_{\text{ref}} \), mathematically this is written as

\[
\rho = \rho_{\text{ref}} \left( 1 - \alpha_V (T - T_{\text{ref}}) \right),
\]

(4.13)

where \( \rho_{\text{ref}} \) is the density of the fluid at the reference temperature \( T_{\text{ref}} \), and \( T \) is the temperature. In our particular case, we have \( T_{\text{ref}} = 106.95[K], \alpha_V = 3.7 \times 10^{-3}[K^{-1}] \), and \( \rho_{\text{ref}} = 430.15[\text{kg m}^{-3}] \). In Figure 4.5 we can observe how this simple linear model captures very well the variations in density, provided that the temperature stays below saturation, i.e., provided that we stay in the liquid phase. Over the whole range of temperatures, i.e., from \( T = 100[K] \) up to the saturation temperature \( T = 146[K] \), the relative error of the linear approximation is less than 0.5%, which is very accurate considering the simplicity of the model.

All the other properties of the fluid are taken as constant. We consider the properties of methane at temperature \( T_{\text{ref}} = 106.95[K] \) and pressure \( P = 900[kPa] \). The respective values are \( \kappa = 0.1915[W \text{ m}^{-1} \text{ K}^{-1}] \) for the thermal conductivity, \( \mu = 1.3247 \times 10^{-4}[\text{Pa s}] \) for the dynamic viscosity, and \( C_p = 3.4395[\text{J g}^{-1} \text{ K}^{-1}] \) for the specific heat capacity.
4.4 Natural Convection

Figure 4.4: Density of methane as function of temperature, for a constant pressure $P = 900$[kPa]. The data was taken from [58].

Figure 4.5: Experimental data and linearization, for the density of liquid methane as function of temperature, for a constant pressure $P = 900$[kPa]. The experimental data was taken from [58].
<table>
<thead>
<tr>
<th>Quantity</th>
<th>Symbol</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>storage temperature</td>
<td>$T_{\text{ref}}$</td>
<td>106.95</td>
<td>[K]</td>
</tr>
<tr>
<td>reference density</td>
<td>$\rho_{\text{ref}}$</td>
<td>430.15</td>
<td>[kg m$^{-3}$]</td>
</tr>
<tr>
<td>volumetric expansion coefficient</td>
<td>$\alpha_V$</td>
<td>$3.7 \times 10^{-3}$</td>
<td>[K$^{-1}$]</td>
</tr>
<tr>
<td>thermal conductivity</td>
<td>$\kappa$</td>
<td>0.1915</td>
<td>[W m$^{-1}$ K$^{-1}$]</td>
</tr>
<tr>
<td>dynamic viscosity</td>
<td>$\mu$</td>
<td>$1.3247 \times 10^{-4}$</td>
<td>[Pa s]</td>
</tr>
<tr>
<td>specific heat capacity</td>
<td>$C_p$</td>
<td>3.4395</td>
<td>[J g$^{-1}$ K$^{-1}$]</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Symbol</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>environment temperature</td>
<td>$T_e$</td>
<td>290</td>
<td>[K]</td>
</tr>
<tr>
<td>wall heat transfer coefficient</td>
<td>$\alpha_w$</td>
<td>0.01 - 0.1</td>
<td>[W m$^{-2}$ K$^{-1}$]</td>
</tr>
</tbody>
</table>

Table 4.1: Properties of methane at $p = 900$[kPa] and $T = 106.95$[K], and wall heat transfer properties.

In addition to the properties of methane, we require the thermal properties of the wall. We consider a constant effective heat transfer coefficient for the wall $\alpha_w$. Depending on the kind of insulation, the heat transfer coefficient of transfer pipes for liquefied gases might vary from $\alpha_w = 0.01$[W m$^{-2}$ K$^{-1}$], for heavily insulated pipes up to $\alpha_w = 2.3$[kW m$^{-2}$ K$^{-1}$] for non-insulated pipes [33]. We concentrate on the case of well-insulated pipes and consider two wall heat transfer coefficients, namely $\alpha_w = 0.01$[W m$^{-2}$ K$^{-1}$] and $\alpha_w = 0.1$[W m$^{-2}$ K$^{-1}$]. The temperature of the environment is taken to be $T_e = 290$[K]. For convenience, the properties of LNG and for the wall heat transfer are summarized in Table 4.1.

### 4.5 Modeling of a Thermosyphon Loop

The flow in a thermosyphon is driven by buoyancy forces which are the result of the density differences along the pipe. However, even though the density is not constant, the density variations can be neglected in the momentum equation, except when they appear multiplied by the gravitational acceleration $g$. In other words, the flow is regarded as incompressible with respect to a reference density $\rho_{\text{ref}}$, and the buoyancy effects are taken into account via a volume force term, which is directed in the opposite direction of the gravitational force $g$. The equations describing such a fluid are known as the Boussinesq approximation to the Navier-Stokes equations, and their derivation was presented in Section 2.3.
4.5 Modeling of a Thermosyphon Loop

4.5.1 Geometry and Governing Equations

We will use the Boussinesq approximation to describe the flow along the thermosyphon loop depicted in Figure 4.3. The quasi-incompressibility assumption applies because the high storage pressure allows the temperature to be well below saturation. Since the cryogenic storage tank has very heavy insulation, most of the motion takes place along the thermosyphon return line. This line consists of two bends and a vertical section. The bends are much shorter than the vertical section, and therefore, we neglect their influence, and simply model the vertical section of the thermosyphon, denoted by $\Omega$ in Figure 4.3.

The geometry which we consider is axially symmetric and periodic and the main goal of the model is to determine the effects of wall-shape on the performance of the thermosyphon. The family of pipes which we consider are depicted in Figure 4.3. The geometry is axially symmetric and periodic with period $L$. Each period consists of two parts, an expansion region of length $E$, and a contraction region of length $C$. The shape of the expansion region is half a period of a sinusoidal shape with period $2E$ and amplitude $a$. The contraction region is half a period of a sinusoidal shape with period $2C$ and amplitude $a$. By construction, the curve describing the wall-shape is continuously differentiable. The constant $D$ denotes the inner diameter of the corrugated pipe and $L_p$ the total length of the pipe. All the simulations presented in this section consider a geometry with $D = 0.02$[m], $L = 5$[m], and $L_p = 0.75$[m]. We also introduce the dimensionless parameter $L_e := E/L$, which measures the proportion of expansion. We want to characterize the effects of the amplitude $a$, and the proportion of expansion $L_e$, on the flow and temperature fields. In addition, we also want to determine the optimal values for these parameters.

We model the flow in the thermosyphon via the Boussinesq approximation. To be more specific, we consider the system of equations describing natural convection which was introduced in Section 2.4.2. Since the geometry under consideration is axially symmetric and the center-line is aligned with the direction of gravity (see Figure 4.3), we can expect the flow to be axially symmetric as well. Furthermore, assuming steady state conditions, the system of equations describing natural convection (2.41) in dimensional form in cylindrical coordinates (see Appendix B eq. B.13) reads

\begin{align}
\partial_x u_x + \partial_r u_r + \frac{1}{r} u_r &= 0, \\
\nu u_x \partial_x u_x + u_r \partial_r u_x &= \nu \left( \partial_{xx} u_x + \partial_{rr} u_x + \frac{1}{r} \partial_r u_x \right) + g \alpha (T - T_{ref}) - \frac{1}{\rho_{ref}} \partial_x p, \\
\nu u_x \partial_x u_r + u_r \partial_r u_r &= \nu \left( \partial_{xx} u_r + \partial_{rr} u_r + \frac{1}{r} \partial_r u_r - \frac{1}{r^2} u_r \right) - \frac{1}{\rho_{ref}} \partial_r p, \\
\rho_{ref} C_p (u_x \partial_x T + u_r \partial_r T) &= \kappa \left( \partial_{rr} T + \frac{1}{r} \partial_r T + \partial_{xx} T \right),
\end{align}

(4.14a) (4.14b) (4.14c) (4.14d)
where the corresponding variables are the axial coordinate \(x\), the radial coordinate \(r\), the axial velocity \(u_x\), the radial velocity \(u_r\), the pressure \(p\) and the temperature \(T\). As it was mentioned in the previous section, \(\rho_{\text{ref}}\) and \(\alpha_V\) are the density and coefficient of volume expansion evaluated at the reference temperature \(T_{\text{ref}}\), respectively. The pressure \(p\) in the equations is the deviation from the hydrostatic pressure of a fluid with density \(\rho_{\text{ref}}\) (see Section 2.3 for more details). The constant \(\nu = \mu/\rho_{\text{ref}}\) is the kinematic viscosity of methane and \(g = 9.806 \text{[m s}^{-2}\text{]}\) is the acceleration of gravity. Since the center-line of the thermosyphon pipeline is assumed to be aligned with the gravitational force, the velocity field is expected to be axially symmetric and therefore, the azimuthal terms do not appear in the equations.

### 4.5.2 Boundary Conditions

The boundary conditions which we consider to model natural convection inside the thermosyphon loop are the following.

1) The flow and temperature fields are axially symmetric

\[
\begin{align*}
u_r &= 0, \quad \partial_r u_x = 0, \quad \partial_r T = 0 \quad \text{at} \quad r = 0. 
\end{align*}
\] (4.15)

2) At the inlet \(\Gamma_{\text{in}}\) and outlet \(\Gamma_{\text{out}}\), we impose the free flow boundary condition for the velocity, also known as the “do nothing” boundary condition \([44, 96]\). In addition, since the system forms a closed loop, the pressure change around the loop should add up zero. Since \(p\) denotes the deviation from the hydrostatic pressure field within the tank, we therefore have

\[
\mu \left( \nabla u + (\nabla u)^T \right) n = 0, \quad p = 0, \quad \text{at} \quad \Gamma_{\text{in}}, \Gamma_{\text{out}}. 
\] (4.16)

3) The temperature at the inlet is uniform and equal to the temperature inside the tank, and at the exit, the heat is carried out mostly due to convection, i.e.,

\[
T = T_{\text{ref}} \quad \text{at} \quad \Gamma_{\text{in}}, \quad \partial_x T = 0 \quad \text{at} \quad \Gamma_{\text{out}}. 
\] (4.17)

4) At the wall of the pipe, a no-slip condition holds for the velocity field, and the normal heat flux is proportional to the difference of the ambient temperature and the local temperature of the fluid at the wall of the pipe. This reads

\[
u = 0, \quad -n \cdot \nabla (-kT) = \alpha_w(T_e - T) \quad \text{at} \quad \Gamma, 
\] (4.18)

where \(n\) denotes the outer unit normal vector to the surface \(\Gamma\), and \(\alpha_w\) is the heat transfer coefficient of the pipe wall.
4.5.3 Numerical Methodology

The system of equations (4.14) introduced in the previous section is solved with a mixed finite element model, with Lagrange $P_2$ and $P_1$-elements, for the velocity field and the pressure, respectively. The temperature is discretized with Lagrange $P_2$-elements. The order of approximation of the pressure is chosen to be one order less than the velocity, in order to avoid an overdetermined discrete system of equations [97]. The discrete non-linear system of equations is solved using Newton iteration. The system of equations is solved with the code COMSOL Multphysics [22]. After solving numerically the discrete system of equations, in a post-processing step, we compute the volumetric flow rate $Q$, and the maximum temperature attained inside the thermosyphon $T_{\text{max}}$ in the following way

$$Q = \int_{\Gamma_{in}} u_x dA, \quad T_{\text{max}} = \max_{x \in \Omega} T(x). \quad (4.19)$$

Based on the volumetric flow rate $Q$, we obtain the average velocity $U_x$, and the Reynolds number $Re$ in the following way.

$$U_x = \frac{4Q}{\pi D^2}, \quad Re = \frac{\rho_{\text{ref}} U_x D}{\mu}. \quad (4.20)$$

We also compute the total amount of heat transferred to the fluid $H$, the average heat flux at the wall $q_w$ and the modified Grashoff number $Gr_{\text{mod}}$ [105], as follows

$$H = \int_{\Gamma} -k \nabla T \cdot \mathbf{n} \, dS, \quad q_w = \frac{H}{|\Gamma|}, \quad Gr_{\text{mod}} = \frac{\rho_{\text{ref}}^2 g \alpha \nu D^4 q_w}{\mu^2 \kappa}, \quad (4.21)$$

where $|\Gamma|$ denotes the surface area of the wall of the pipe.

In particular, the maximum temperature $T_{\text{max}}$ attained in the domain is an important design variable because the temperature should be kept below saturation everywhere along the thermosyphon, in order to maintain the liquid phase throughout the return line and prevent malfunctioning [95].

In order to guarantee grid independent solutions for each of the geometries considered here, we implemented a routine which recursively refines the mesh, until the relative change in the computed volumetric flow rate $Q$ is less than 0.1%. In Figure 4.6, we can see a close up of the initial and the refined mesh nearby the corrugation of a geometry with parameters $L = 5[\text{mm}]$, $a = 2[\text{mm}]$, and $L_c = 0.5$. As we can observe, the mesh density increases inside the corrugation in order to correctly solve the vortex which develops inside. The relative variations in the computed values of $Q$, and $H$ (total heat transfer), can be observed in Figure 4.7. A solution with a relative variation smaller than 0.1% for $Q$ and $H$, is obtained with a mesh consisting of about $4 \times 10^4$ mesh points. We also notice that by inheritance, the variables $Re$ and $Gr_{\text{mod}}$ have the same accuracy.
Figure 4.6: Close up of the initial and refined mesh nearby a corrugation of a geometry with $L = 5\text{mm}$, $a = 2\text{mm}$, and $L_c = 0.5$.

Figure 4.7: Relative variation in the Reynolds $Re$ number and total heat transfer $H$, as function of the number of mesh points. The geometry consider here has $L = 5[\text{mm}]$, $a = 2[\text{mm}]$, and $L_c = 0.5$ (see Figure 4.3).
4.6 Effects of Wall-Shape

In order to have an idea of the flow and temperature fields which develop inside the thermosyphon, we discuss first the case of a straight pipe, i.e., \( a = 0 \). Even for this simple geometry, the velocity and temperature distributions show an interesting behavior. In Figure 4.8, we have plotted the axial velocity \( u_x \), and the temperature difference \( T - T_{ref} \), for a pipe with a wall heat transfer coefficient of \( \alpha_w = 0.01 \text{[W m}^{-2} \text{K]} \). In the figure, the geometry has been scaled in order to show the patterns over the whole domain \( \Omega \) (see Figure 4.3), i.e., over the total length of the thermosyphon line \( L_p = 0.75 \text{[m]} \) and the radius \( R_w = 0.01 \text{[m]} \). At the entrance of the pipe, a high velocity region shows next to the wall and as one goes further in the axial direction, the high velocity region moves towards the center line. As one comes close to the exit of the pipe, the velocity profile is distorted and the maximum velocity shifts from the center line towards the wall of the pipe, showing a typical feature of natural convecting flow \([105]\). The light and dark areas denote cold and hot regions respectively. As we see, a thermal boundary layer develops along the wall of the pipe. The Reynolds number for this wall heat transfer coefficient was \( Re \approx 638 \).

When we increase the heat transfer coefficient of the wall from \( \alpha_w = 0.01 \text{[W m}^{-2} \text{K]} \) to \( \alpha_w = 0.1 \text{[W m}^{-2} \text{K}^{-1}] \), we of course expect a higher velocity inside the thermosyphon. This is reflected in the Reynold snmber, which in this case increases to \( Re = 1719 \). In Figure 4.9 we can observe the velocity and temperature fields for this higher wall heat transfer coefficient. As we see, the temperature field now shows a steeper thermal boundary layer at the wall of the pipe, due to the higher velocity attained inside the thermosyphon. The plot of the axial velocity field shows that near the exit of the pipe, the high velocity region has shifted a bit further towards the wall of the pipe and the difference is also more pronounced, than in the case with the smaller wall heat transfer coefficient.

This example shows that the entrance effects are very important for the thermosyphon loop. Now we proceed to study the case when the wall of the pipe is not straight, i.e., for a corrugated pipe.

4.6.1 Influence of Corrugation Size \( a \)

Now we study the case when the thermosyphon return line consists of a corrugated pipe. We start by studying the effects of the amplitude \( a \) on the flow. In Figure 4.10 we have plotted the maximum increase in temperature \( T_{max} - T_{ref} \) and the Reynolds number for a corrugated pipe depicted as in Figure 4.3. The values of the geometrical parameters in this case are \( L = 5 \text{[mm]}, L_c = 0.5 \) and \( \alpha_w = 0.1 \text{[W m}^{-2} \text{K}^{-1}] \). The amplitude of the corrugation \( a \) is allowed to vary in order to see how it affects the the flow. In the case
Figure 4.8: Axial velocity and temperature fields in a straight pipe (i.e. $a = 0$) with wall heat transfer coefficient $\alpha_w = 0.01\,[\text{W} \text{m}^{-2} \text{K}^{-1}]$.

Figure 4.9: Axial velocity and temperature fields in a straight pipe (i.e. $a = 0$) with wall heat transfer coefficient $\alpha_w = 0.1\,[\text{W} \text{m}^{-2} \text{K}^{-1}]$. 
of a straight pipe, i.e. $a = 0\,[\text{m}]$, the increment in temperature is about $T_{\text{max}} - T_{\text{ref}} \approx 0.049\,[\text{K}]$ and the Reynolds number $Re \approx 638$. When we increase the amplitude of the corrugation, we obtain an increase in temperature and in the Reynolds number. For instance, when $a = 2.8\,[\text{mm}]$, the Reynolds number reaches a value of $Re \approx 862$, and the increase in temperature is $T_{\text{max}} - T_{\text{ref}} \approx 0.1\,[\text{K}]$. From Figure 4.10, we can also notice that while the Reynolds number increases almost linearly, the temperature does not. In fact, the slope of the temperature curve increases steadily with $a$ up to $a \leq 0.6\,[\text{mm}]$. Then the slope becomes constant and the temperature starts to grow linearly till $a \approx 1.4\,[\text{mm}]$. As we increase $a$ further, we can still observe another change in the slope of the curve at $a = 2.8\,[\text{mm}]$. To show this more clearly, we have added two auxiliary straight lines (one dotted and one dashed) for comparison.

This behavior of the increase in temperature can be explained by looking at the flow streamlines, which we have plotted in Figure 4.11. For instance, when $a = 0.6\,[\text{mm}]$ (see Figure 4.11(a)) the flow follows the wall and the increase in temperature is just caused by the fact that the surface area (and therefore, also the heat transfer) increases with $a$. When we reach $a = 1\,[\text{mm}]$ the temperature starts to show a linear behavior, this is associated with the appearance of a vortex inside the corrugations. For instance, when $a = 1.4\,[\text{mm}]$, we can see that a vortex has developed inside the corrugations (see Figure 4.11(b)). The temperature continues to increase in a linear way until at $a \approx 2.7\,[\text{mm}]$ we can observe another change in the slope of the curve. This change takes place due to the appearance of a second vortex inside the corrugation. This can be confirmed in Figure 4.11(c), where for an amplitude of $a = 2.8\,[\text{mm}]$ we can already distinguish a second vortex developing in the deep region of the corrugation.

In practical terms, the previous discussion suggests that the maximum temperature inside the thermosyphon always increases with increasing amplitude $a$. In other words, for two pipes with the same inner diameter $D$, the maximum temperature in a straight thermosyphon line is always smaller than the maximum temperature inside a corrugated pipe. However, in order to have a flexible pipe line, and therefore benefit from the operational point of view, it is necessary to have a minimum amplitude size for the corrugation. Therefore, in a practical situation, one should balance the increase in temperature inside the thermosyphon and the desired flexibility of the pipe line in order to choose an optimal design value for the amplitude $a$.

### 4.6.2 Influence of Proportion of Expansion $L_e$

Now we study the influence of the parameter $L_e$. In fact, we will see that for a given amplitude $a$, it is possible to reduce the maximum temperature inside the thermosyphon without having to compromise the flexibility of the line. When we modify the parameter $L_e$, the lengths of expansion and contraction of the pipe change. A value
Figure 4.10: Maximum increment in temperature $T_{\text{max}} - T_{\text{ref}}$ and Reynolds number Re as function of the amplitude $a$, for a wall heat transfer coefficient $\alpha_w = 0.1\,[\text{W m}^{-2}\text{K}^{-1}]$.

Figure 4.11: Flow streamlines for a sinusoidal pipe with wall heat transfer coefficient $\alpha_w = 0.1\,[\text{W m}^{-2}\text{K}^{-1}]$, and three different amplitudes, namely $a = 0.6\,[\text{mm}]$ in Figure 4.11(a), $a = 1.4\,[\text{mm}]$ in Figure 4.11(b) and $a = 2.8\,[\text{mm}]$ in Figure 4.11(c). The period of the sinusoidal pipe is $L = 5\,[\text{mm}]$, the unit of the axes are expressed in meters and the flow direction is upwards.
4.6 Effects of Wall-Shape

$L_e = 0.5$ correspond to a symmetric shape, while a value of $L_e = 0.7$ corresponds to a geometry in which the expansion length constitutes 70% of the period $L$, i.e., to a geometry with large expansion length and short contraction length.

Now we proceed to study the effects of the parameter $L_e$. In Figures 4.12(a) and 4.12(b), we can see a close-up of the temperate fields $T - T_{\text{ref}}$ nearby the corrugations for two geometries, one with $L_e = 0.55$ and another one with $L_e = 0.7$. All the other parameters were the same in both cases, namely $\alpha_w = 0.1[W \text{ m}^{-2} \text{ K}^{-1}]$, $L = 5[\text{mm}]$, $a = 2[\text{mm}]$ and $D = 2[\text{cm}]$. When $L_e = 0.7$, the heat transfer due to convection is reduced by the sharp contraction region, and this causes the temperature to rise by $0.47[\text{K}]$. On the other hand, when $L_e = 0.55$, the fluid inside the cavity is able to exchange heat with the main cold stream in a better way, which in turn reflects on the maximum increase in temperature along the whole thermosyphon, yielding the slightly smaller value of $T_{\text{max}} - T_{\text{ref}} = 0.464[\text{K}]$.

The role of the parameter $L_e$ becomes more clear by looking at Figure 4.13. In this figure we have plotted the maximum increment of the temperature $T_{\text{max}} - T_{\text{ref}}$ inside the thermosyphon, for a set of corrugated pipes with period $L = 5[\text{mm}]$ and amplitude $a = 2[\text{mm}]$. The proportion parameter $L_e$ was allowed to vary from 0.25 up to 0.75. The solid line and left hand $y$-axis show the case for a heat transfer coefficient of $\alpha_w = 0.01[W \text{ m}^{-2} \text{ K}^{-1}]$. The dashed line and right hand $y$-axis correspond to a heat transfer coefficient of $\alpha_w = 0.1[W \text{ m}^{-2} \text{ K}^{-1}]$. Already in the case when $\alpha_w = 0.01[W \text{ m}^{-2} \text{ K}^{-1}]$, it is possible to notice certain asymmetry in the curve. The minimum increment in temperature appears to be attained for an asymmetric geometry, i.e. for $L_e > 0.5$. However, the deviation is quite small and it is a bit difficult to notice, we have added an auxiliary line to the figure to show this more clearly.

The asymmetric effect of the parameter $L_e$ becomes much more evident when we increase the heat transfer coefficient of the wall $\alpha_w$. The dashed line in Figure 4.13, shows the maximum increase in temperature $T_{\text{max}} - T_{\text{ref}}$ as a function of the parameter $L_e$, for a wall heat transfer coefficient of $\alpha_w = 0.1[W \text{ m}^{-2} \text{ K}^{-1}]$. The first effect that we can notice, is that the increase in temperature $T_{\text{max}} - T_{\text{ref}}$ becomes more sensitive to the parameter $L_e$. For example, for $\alpha_w = 0.01[W \text{ m}^{-2} \text{ K}^{-1}]$, $T_{\text{max}} - T_{\text{ref}}$ varies between $0.0815[\text{K}]$ and $0.085[\text{K}]$ in the range $0.25 \leq L_e \leq 0.75$, i.e., a maximum variation of about $0.0035[\text{K}]$. On the other hand, when $\alpha_w = 0.1[W \text{ m}^{-2} \text{ K}^{-1}]$, $T_{\text{max}} - T_{\text{ref}}$ varies between $0.463[\text{K}]$ and $0.471[\text{K}]$, in the smaller range $0.42 \leq L_e \leq 0.7$, i.e. a maximum variation of about $0.08[\text{K}]$. The second observation that we can make, is that now the asymmetric behavior becomes more prominent and in this case, we can clearly see that the minimum increase in temperature, is attained for a value around $L_e \approx 0.57$, i.e., a geometry where the length of expansion is larger than the length of contraction.

In other words, the parameter $L_e$ gives us the possibility of reducing the temperature increase inside the thermosyphon, without having to reduce the amplitude of the cor-
rugation, which is the main feature responsible for the flexibility of the pipe line, hence keeping the operational advantages of a corrugated line and reducing the temperature increase due to the corrugations. The role of the parameter $L_e$ becomes more important when the heat transfer at the wall increases. Also, we can expect the parameter $L_e$ to play a more important role for corrugations with larger periods $L$. This is suggested by the results which we obtained for isothermal laminar flow in Section 3.5. In this section we saw that the parameter $L_e$ plays an important role specially in the case of slowly varying geometries, i.e., in the case of a large period $L$. Overall, we see that it is possible to optimize the wall-shape in order to reduce the increase in temperature inside the thermosyphon, by tuning the parameter $L_e$.

4.7 Remarks on Non-Isothermal Laminar Flow

In the present chapter we have addressed the problem of non-isothermal laminar flow in corrugated pipes. We discussed the cases of forced and natural convection. In the case of forced convection we presented an efficient way to solve the problem with constant heat flux. We were able to reduce the computational domain to just one period by applying a periodicity type argument. Since the problem posed no extra additional challenge in comparison to the case of isothermal laminar flow, we turned our attention to the case of natural convection.
4.7 Remarks on Non-Isothermal Laminar Flow

In the second part we discussed this case by addressing a real practical application involving natural convection. We presented a model for a single phase thermosyphon loop featuring corrugated pipes. By means of using an adaptive meshing technique we were able to predict the flow and temperature fields with enough detail to capture the effects of wall-shape. We considered a set of asymmetric sinusoidal pipes and showed that an increase in the amplitude implies an increase in flow rate and in temperature. The appearance of vortices inside the corrugation induces a change in the slope of the temperature as function of the amplitude $a$. The heat transfer in corrugated pipes is larger than in straight pipes and one should take this into account when considering to change a straight thermosyphon line to a corrugated one.

In addition, we showed that it is possible to reduce the maximum temperature inside the thermosyphon by tuning the length of expansion and contraction of the pipe. In particular, we showed that for a given corrugation amplitude $a$, there is an optimal value for the parameter $L_c$. In practical terms, this means that we do not need to compromise the flexibility of the corrugated line (i.e. we can keep the same amplitude $a$, roughly speaking) in order to reduce the maximum temperature inside the thermosyphon. The design parameter $L_c$ becomes more important when the wall heat transfer coefficient $\alpha_w$ increases and also when the the period of the corrugation $L$ increases. In summary, shape design appears promising for reducing the temperature increase inside the thermosyphon and consequently, for preventing malfunctioning due to bubble formation.
Chapter 5

Homogenization of the Boussinesq Equations

In the present chapter we introduce the so-called method of homogenization in order to be able to tackle more efficiently the problem of natural convection in a vertical corrugated pipe, introduced in the previous chapter. In practical terms, the homogenization method allows us to replace the complex boundary of the corrugated wall by a simple flat boundary. The boundary conditions at this artificial flat boundary are derived in such a way that the solution on the simplified domain closely approximates the solution on the original domain. With the solution on the simplified domain it is possible to compute the drag, the flow rate and heat transfer, but since the details of the geometry are no longer required, the problem can be solved in a much more efficient way as we will see further on in this chapter.

5.1 Idea of Homogenization

Originally, the homogenization technique was introduced to describe the behavior of composite materials [19,35], and it is also very common in the theory of porous media. Composite materials are characterized by having two or more finely mixed constituents, in other words, the material is heterogeneous, however, from the macroscopic point of view, the composite material looks simply like a homogeneous material. The original idea of homogenization is to derive macroscopic properties of the composite by taking into account the properties of the microscopic structure, hence the name homogenization.
We will use an analogous idea for treating the problem of natural convection in a corrugated pipe. In our particular case, the corrugations play the role of the heterogeneities. The corrugations are located at the wall of the pipe and are not distributed over the whole domain, which contrast with the traditional homogenization problems, such as for composite materials and porous media [19]. Thus, we want to apply a homogenization procedure to the boundary of the domain. In the homogenization procedure we will attempt to replace the original boundary (including the corrugations) by a simpler homogenized boundary (a simple straight wall), in which we will impose effective boundary conditions. The so-called effective boundary conditions have the purpose of taking into account the effects of the corrugations on the velocity and temperature fields away from the wall, without having to find the solution inside the corrugations. The advantage of using such an effective boundary condition, is that the homogenized domain is much simpler than the original one, and therefore it is much easier to solve the system of equations in the homogenized domain. For example, the generation of adequate meshes for solving the equations becomes straightforward, and to obtain a mesh independent solution one requires much less elements than when solving in the original geometry. In fact, to numerically solve a problem in a domain with scales which differ by several orders of magnitude is almost impossible and sometimes just generating an initial mesh for such a geometry is not an easy task [63].

5.2 Mathematical Model

The geometry which we consider is a periodic axially symmetric corrugated pipe, depicted as in Figure 5.1. As it can be observed from this figure, the wall of the pipe consist of small periodic corrugations along the axial direction. The main assumption on which we will base the following derivations is that the typical size of the corrugations is small. Generally speaking, this means that the flow in the main stream (away from the wall) should not be strongly affected by the presence of the corrugated wall.

5.2.1 Natural Convection Equations

The phenomenon which we want to model is natural convective flow in a corrugated pipe. We represent the geometry of the corrugated pipe by an axially symmetric geometry. As in Section 4.5, we consider the case when the center-line of the corrugated pipe is aligned with the direction of gravity, and hence it is plausible to assume the solution to be axially symmetric. Considering the description of axisymmetric geometries in Section 2.5.2, an axisymmetric solution means that we only need to solve the corresponding system of equations on one half of a cross-section, i.e., at the region \( \theta = 0 \) for instance. As we discussed in Chapter 2, natural convecting flows can be described by
the Boussinesq approximation. Omitting the superscript $\star$ (for notational convenience), the *dimensionless Boussinesq system* (2.41) in steady state reads

\[ \nabla \cdot \mathbf{u} = 0, \]
\[ \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u} - \frac{g}{g} T, \]
\[ \mathbf{u} \cdot \nabla T = \frac{1}{Pe} \nabla^2 T, \]

where $\mathbf{u}$, $p$ and $T$, are the dimensionless velocity, pressure and temperature, respectively (see Section 2.4). The characteristic length considered is $l_0 = D$, where $D$ corresponds to the minimum (dimensional) diameter (see Section 2.5.3). The Reynolds and Péclet number in the equations take the form $Re = U_x D/\nu$, and $Pe = Re Pr$, where $Pr = C_p \mu/\kappa$ is the Prandtl number (see (2.33)). Since the flow direction is opposite to the acceleration of gravity we have $g = -ge_x$ and hence we can replace the buoyancy term of (5.1b) by $e_x T$.

### 5.2.2 Geometry and Boundary Conditions

The system of equations (5.1) holds in the (dimensionless) domain $\Omega_\epsilon$ depicted as in Figure 5.1. Here $L_p$ denotes the dimensionless length of the pipe. Since the characteristics length used to non-dimensionalize the system is the minimum diameter $D$, with this choice, the minimum value of the dimensionless radius $R_w = R_w(x)$ becomes 1/2.
as shown in Figure 5.1. Without loss of generality we consider \( R_w(0) = 1/2 \). We assume both, the amplitude and period of the corrugations of the wall to be of order \( \epsilon \), with \( 0 < \epsilon \ll 1 < L_p \). This assumption means that the dimensionless radius \( R_w \) can be written as

\[
R_w(x) = R_w^0 + \epsilon R_{\text{in}} \left( \frac{x}{\epsilon} \right),
\]

(5.2)

where \( R_w^0 = \frac{1}{2} \). The function \( R_{\text{in}} \) describing the corrugations, is \( O(1) \) and it is periodic with respect to the variable \( x/\epsilon \). By writing \( R_w \) as in (5.2), we have conveniently represented the assumption that the period and the amplitude of the corrugations are \( O(\epsilon) \).

The system of equations (5.1) is complemented by appropriate conditions along the boundary of the domain \( \partial \Omega_\epsilon \). We split the boundary of the domain as follows (see Figure 5.1)

\[
\partial \Omega_\epsilon = \Gamma_{\text{bot}} \cup \Gamma_{\text{top}} \cup \Gamma_c \cup \Gamma_\epsilon,
\]

(5.3)

where \( \Gamma_{\text{bot}} \), \( \Gamma_{\text{top}} \), \( \Gamma_c \) and \( \Gamma_\epsilon \) denote the inlet, outlet, center line and wall of the pipe, respectively. To be more precise we define these regions as follows

\[
\Gamma_{\text{bot}} := \left\{ r \mathbf{e}_r \in \mathbb{R}^2 \mid 0 \leq r \leq R_w(0) \right\},
\]

(5.4a)

\[
\Gamma_{\text{top}} := \left\{ r \mathbf{e}_r + L_p \mathbf{e}_x \in \mathbb{R}^2 \mid 0 \leq r \leq R_w(0) \right\},
\]

(5.4b)

\[
\Gamma_c := \left\{ x \mathbf{e}_x \in \mathbb{R}^2 \mid 0 \leq x \leq L_p \right\},
\]

(5.4c)

\[
\Gamma_\epsilon := \left\{ R_w(x) \mathbf{e}_r + x \mathbf{e}_x \in \mathbb{R}^2 \mid 0 \leq x \leq L_p \right\}.
\]

(5.4d)

At the center line \( \Gamma_c \), we consider an axial symmetry boundary condition, namely

\[
u_r = 0, \quad \partial_r u_x = 0, \quad \partial_r T = 0 \quad \text{at} \quad \Gamma_c.
\]

(5.5)

At the wall of the pipe we consider a boundary condition (of Robin type) for the heat flux, and the no-slip boundary condition for the velocity field, i.e.,

\[
n \cdot \nabla T = f_q, \quad u = 0 \quad \text{at} \quad \Gamma_\epsilon,
\]

(5.6)

where \( n \) is the outer normal vector, \( f_q \) is a function describing the heat transfer at the wall of the pipe, and might be dependent on the position \( x \) and on the local temperature \( T \). As an example we consider the boundary condition (4.18), which we used in Section 4.5.2 to model a thermosyphon loop. For this particular case, if we take \( \Delta T = T_e - T_{\text{ref}} \) as the characteristic temperature difference for the non-dimensionalization (see (2.30)), then the boundary condition in dimensionless variables reads (again omitting the su-
perscripts \( \star \) \]

\[
n \cdot \nabla T = \text{Nu}_D (1 - T),
\]

(5.7)

where \( \text{Nu}_D := \alpha_w D / \kappa \) is the Nusselt number based on the reference diameter \( D \), the wall heat transfer coefficient \( \alpha_w \), and the thermal conductivity of the fluid \( \kappa \). We will use this particular boundary condition for a numerical study in a later section. However, for our formal homogenization, we will consider the more general boundary condition (5.6). The boundary conditions at the inlet \( \Gamma_{\text{bot}} \) and at the outlet \( \Gamma_{\text{top}} \) can be written in dimensionless form as well. However, as we will see, the specific form of the boundary conditions at the inlet and at the outlet is irrelevant as far as the upscaling procedure of the boundary \( \Gamma_\epsilon \) is concerned.

### 5.3 Formal Homogenization

The main goal of boundary homogenization is to replace the corrugated boundary \( \Gamma_\epsilon \), by a much simpler one with an effective boundary condition. Our main assumption is that the size of the corrugations is small compared to the diameter of the pipe. We formalized this assumption in (5.2). In contrast to the slowly varying case treated in Section 3.4.1, now we allow for corrugations with amplitude of the same order as the period. As the parameter \( \epsilon \) becomes smaller, the number of corrugations along \( \Gamma_\epsilon \) increases, while the amplitude of the corrugations decreases, thus keeping the ratio between the period and the amplitude of the pipe. Since \( R_{\text{w}}^\infty = O(1) \), when we take the limit \( \epsilon \to 0 \), the corrugated boundary converges to the flat boundary \( \Gamma_0 := \{ R_w^\infty e_r + x e_x \in \mathbb{R}^2 \mid 0 \leq x \leq L_p \} \). In a similar manner we have that \( \Omega_\epsilon \) converges to a rectangular domain, which we will denote by \( \Omega_0 := \{ r e_r + x e_x \in \mathbb{R}^2 \mid 0 \leq x \leq L_p, \ 0 \leq r \leq R_w^0 \} \).

The boundary \( \Gamma_0 \) and the domain \( \Omega_0 \) will be our homogenized boundary and domain, respectively. The goal of the formal homogenization procedure that follows is to find a system of equations in the simplified domain \( \Omega_0 \), and effective boundary conditions at \( \Gamma_0 \) in such a way that the solutions of the homogenized system are a good approximation to the solution of (5.1) subject to (5.6).

Similar approaches have been applied to the Stokes and Navier-Stokes systems without the energy equation in [1, 2]. In [49] a rigorous derivation of the Navier friction condition (effective boundary condition) is provided for the Navier-Stokes equations. The convection-diffusion equation with imposed velocity field has been treated for a problem involving free boundaries in [54]. In our case, we consider the full steady state Navier-Stokes and energy equations with two way coupling, i.e., via the buoyancy forces in the momentum equations and via the convection term in the temperature equation.
Regarding the boundary conditions, we consider axial symmetry at the center line $\Gamma_c$. At the wall of the pipe described by the wavy boundary $\Gamma_\epsilon$, we consider a general non-homogeneous Neumann boundary condition for the temperature equation, and the no-slip condition for the flow equations. Summarizing, the system to which we will apply our formal homogenization is the following Boussinesq system

\[
\nabla \cdot \mathbf{u} = 0, \quad \text{(5.8a)}
\]
\[
\mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u} + \epsilon \mathbf{e}_x T, \quad \text{in } \Omega_\epsilon \quad \text{(5.8b)}
\]
\[
\mathbf{u} \cdot \nabla T = \frac{1}{Pe} \nabla^2 T, \quad \text{(5.8c)}
\]

subject to the boundary conditions

\[
\mathbf{u} = 0, \quad \mathbf{n} \cdot \nabla T = f_q, \quad \text{on } \Gamma_\epsilon, \quad \text{(5.9a)}
\]
\[
u_r = 0, \quad \partial_r u_x = 0, \quad \partial_r T = 0, \quad \text{on } \Gamma_c. \quad \text{(5.9b)}
\]

In (5.8) we already substituted $\mathbf{g} = -g \mathbf{e}_x$ (since the center line is assumed to be aligned with gravity). In order to obtain the homogenized system of equations and effective boundary conditions, we will use asymptotic analysis. The boundary conditions at $\Gamma_{\text{bot}}$ and $\Gamma_{\text{top}}$ are left unspecified because they do not play an active role in the homogenization procedure. By this we mean that regardless of the specific form of the boundary conditions at $\Gamma_{\text{bot}}$ and $\Gamma_{\text{top}}$, the following homogenization procedure is still applicable.

### 5.3.1 Asymptotic Methods

From physical experience, the solution to the system of equations (5.8) subject to the boundary conditions (5.9), is expected to exhibit a boundary layer for the velocity and temperature fields along the boundary $\Gamma_\epsilon$ [6, 113]. We can observe this behavior in Figure 4.8 from Section 4.6, for example. In addition, the corrugations of the wall $\Gamma_\epsilon$ introduce fast variations in the velocity and temperature fields. These variations are of the same order as the size of the corrugations, i.e., $O(\epsilon)$. In Figure 4.11 we can observe the appearance of vortices, which indeed are comparable in size to the corrugations. Figures 4.12(b) and 4.12(a), show the variations of the temperature field due to the presence of the corrugations. From these figures, we can also observe how the influence of the corrugations becomes less prominent as we move away from the wall.

This brief discussion already provides us with some valuable information. First, we expect the appearance of a boundary layer near the wall $\Gamma_\epsilon$. In addition we expect a multiscale behavior in the axial direction close to the boundary. Asymptotic analysis provides us with analytical techniques to handle both kind of phenomena. Both phenomena show singular behavior and require special methods. In the case of boundary
5.3 Formal Homogenization

layer problems, one needs to decompose the solution domain into an outer and an inner region, and then relate the outer and the inner problems via matching conditions. This method is known as the method of matched asymptotic expansions [64]. Problems with several scales are handled with the method of multiple scales. In this method, the different length scales are considered as independent variables [64]. For the problem at hand, we will need to consider both methods.

We now introduce a boundary layer variable in the radial direction and a fast variable in the axial direction, we will denote them by \( \eta \) and \( \xi \), respectively. The form of the fast variable \( \xi \), in fact already appears in the expression for the radius \( R_w \) (5.2). Therefore we define

\[
\xi := \frac{x}{\epsilon}.
\]  

(5.10)

Since we expect the boundary layer to be located at the wall of the pipe \( \Gamma_\epsilon \), we define

\[
\eta := \frac{r - R_w^0}{\epsilon},
\]

(5.11)

where \( R_w^0 \) shows the location of the boundary layer.

The method of matched asymptotic expansions is based on the splitting of the domain into an outer (away from the wall \( \Gamma_\epsilon \)) and an inner region (close to the wall \( \Gamma_\epsilon \)). We will denote the outer solutions with the superscript “out”, and the inner solutions with the superscript “in”. For example, \( u_{x}^{\text{out}} \) will denote the axial velocity component of the outer solution, and \( T^{\text{in}} \) denotes the temperature of the inner solution. In the outer region it is possible to expand the solution as a regular asymptotic expansion in terms of the original variables. In the inner region one first needs to introduce a stretched variable (boundary layer variable). The solution allows a regular asymptotic expansion in terms of the boundary layer variable. The solutions in the outer and the inner regions are related with each other via the matching conditions [45].

In our particular case we have two outer variables \( x \) and \( r \), one boundary layer variable \( \eta \), and one fast variable \( \xi \). Our boundary layer ansatz is the following. On the one hand, the outer solution does not depend on the boundary layer variable \( \eta \), but still, it might be dependent on the fast variable \( \xi \), i.e.,

\[
u = u^{\text{out}}(r, x, \xi; \epsilon), \quad p = p^{\text{out}}(r, x, \xi; \epsilon), \quad T = T^{\text{out}}(r, x, \xi; \epsilon).
\]  

(5.12)

On the other hand, the solution in the inner region is independent of the outer variable \( r \), i.e.,

\[
u = u^{\text{in}}(\eta, x, \xi; \epsilon), \quad p = p^{\text{in}}(\eta, x, \xi; \epsilon), \quad T = T^{\text{in}}(\eta, x, \xi; \epsilon).
\]  

(5.13)
All the functions in (5.12) and (5.13), are assumed periodic in the variable $\xi$, with period $L^\text{in}$, where $L^\text{in}$ is the period of the local radius $R^\text{in}(\xi)$. The period $L$ of the corrugations in the original geometry can be expressed in terms of $L^\text{in}$ by $L = \epsilon L^\text{in}$.

Both inner as well as the outer solutions contain $\epsilon$ as a parameter. The asymptotic ansatz that we will use to proceed states that the outer and inner velocities, pressures and temperatures, in (5.12) and (5.13), can be expanded in asymptotic series, i.e.,

$$
\begin{align*}
&u^\text{out}_r = \sum_{i=0}^\infty u^\text{out}_{r,i} \epsilon^i, \quad u^\text{out}_x = \sum_{i=0}^\infty u^\text{out}_{x,i} \epsilon^i, \\
&p^\text{out} = \sum_{i=0}^\infty p^\text{out}_i \epsilon^i, \quad T^\text{out} = \sum_{i=0}^\infty T^\text{out}_i \epsilon^i,
\end{align*}
$$

(5.14)

$$
\begin{align*}
&u^\text{in}_r = \sum_{i=0}^\infty u^\text{in}_{r,i} \epsilon^i, \quad u^\text{in}_x = \sum_{i=0}^\infty u^\text{in}_{x,i} \epsilon^i, \\
&p^\text{in} = \sum_{i=0}^\infty p^\text{in}_i \epsilon^i, \quad T^\text{in} = \sum_{i=0}^\infty T^\text{in}_i \epsilon^i,
\end{align*}
$$

(5.15)

where the outer functions $u^\text{out}_{r,i}, u^\text{out}_{x,i}, p^\text{out}_i$ and $T^\text{out}_i$ are independent of $\epsilon$, i.e., they depend on $r, x,$ and $\xi$ only. The inner functions $u^\text{in}_{r,i}, u^\text{in}_{x,i}, p^\text{in}_i$ and $T^\text{in}_i$ are as well independent of $\epsilon$, which means they can only depend on $\eta, x$ and $\xi$.

### 5.3.2 Differentiation Rules

For the next derivations it is convenient to introduce the following calculation rules. First we recall the expressions of the gradient and the divergence operators in cylindrical coordinates (under the assumption of axial symmetry), namely

$$
\nabla = e_r \partial_r + e_x \partial_x, \quad \nabla \cdot \mathbf{u} = \frac{1}{r} \partial_r (ru_r) + \partial_x u_x,
$$

(5.16)

where $\mathbf{u} = e_r u_r + e_x u_x$ is a vector field and $u_r$ and $u_x$ are its components in cylindrical coordinates. We note that the expression for the divergence operator in (5.16) applies only when the azimuthal component of the vector field is 0. Since the outer solutions $u^\text{out}, p^\text{out}$ and $T^\text{out}$ depend on $r, x,$ and $\xi$ (see (5.12)), and using the definition of $\xi$ (5.10), we can apply the chain rule to obtain the differentiation relations $\partial_r T = \partial_T T^\text{out}$, $\partial_x T = \partial_x T^\text{out} + \frac{1}{\epsilon} \partial_\xi T^\text{out}$. The same rules hold for any other scalar field in the outer region, such as the components of the velocity $\mathbf{u}$, or the pressure $p$. These relations also allow us to write the gradient and divergence operators as follows

$$
\begin{align*}
\nabla T &= e_r \partial_r T^\text{out} + e_x \left( \partial_x T^\text{out} + \frac{1}{\epsilon} \partial_\xi T^\text{out} \right), \\
\nabla \cdot \mathbf{u} &= \frac{1}{r} \partial_r (ru_r) + \partial_x u_x + \frac{1}{\epsilon} \partial_\xi u_x.
\end{align*}
$$

(5.17)

Here we showed the form of the gradient and the divergence for the temperature and velocity fields, respectively, but in fact the previous expressions are valid for any other scalar or vector field in the outer region.
In the inner region, we know that $u^{in}$, $p^{in}$ and $T^{in}$ depend on $\eta$, $x$ and $\xi$ (5.13). Thus applying the chain rule and using the definition of $\xi$ and $\eta$ in (5.10) and (5.11), we get $\partial_\eta T = \frac{1}{\epsilon} \partial_\eta T^{in}$ and $\partial_\xi T = \frac{1}{\epsilon} \partial_\xi T^{in}$. The same differentiation identities hold for the other inner solutions. The previous differentiation rules allow us to obtain the following expressions for the gradient and divergence operators

$$\nabla T = e_r \frac{1}{\epsilon} \partial_\eta T^{in} + e_x \left( \partial_x T^{in} + \frac{1}{\epsilon} \partial_\xi T^{in} \right),$$

$$\nabla \cdot u = \frac{1}{\epsilon} \partial_\eta u^{in}_r + \frac{1}{R_e + \epsilon \eta} u^{in}_r + \partial_x u^{in}_x + \frac{1}{\epsilon} \partial_\xi u^{in}_x.$$

5.4 Homogenization of the Temperature Equation

In order to obtain effective boundary conditions for the temperature equation (5.8c), we will expand the temperature and velocity fields $T$ and $u$ in the outer and in the inner region in asymptotic series with respect to $\epsilon$. The equation in the outer region requires to consider terms up to $O(1)$, and the equation in the inner region up to $O\left(\frac{1}{\epsilon}\right)$.

5.4.1 Outer Equation

We now proceed to apply formal homogenization to the temperature equation (5.8c). We start by handling the outer solution of the temperature equation. By using the calculations rules (5.17), we first obtain the following expressions for the convective and diffusive terms in (5.8c)

$$\nabla \cdot \left( \nabla T^{out} \right) = \frac{1}{r} \partial_r \left( r \partial_r T^{out} \right) + \partial_x T^{out} + \frac{2}{\epsilon} \partial_\xi T^{out} + \frac{1}{\epsilon^2} \partial_{\xi \xi} T^{out},$$

$$u^{out} \cdot \nabla T^{out} = u^{out}_x \partial_x T^{out} + \frac{1}{\epsilon} u^{out}_x \partial_\xi T^{out} + u^{out}_r \partial_r T^{out}.$$

Using the previous expressions, expanding in asymptotic series as in (5.14) and collecting terms with respect to their order in $\epsilon$, we obtain the following hierarchical set of
Homogenization of the Boussinesq Equations

\[ \mathcal{O} \left( \frac{1}{\epsilon} \right) : \partial_{\xi} T_{0}^{\text{out}} = 0, \]  
(5.22a)

\[ \mathcal{O} \left( \frac{1}{\epsilon} \right) : u_{\xi,0}^{\text{out}} \partial_{\xi} T_{0}^{\text{out}} = \frac{1}{Pe} \left( 2 \partial_{x} T_{0}^{\text{out}} + \partial_{\xi} T_{1}^{\text{out}} \right), \]  
(5.22b)

\[ \mathcal{O} \left( \epsilon \right) : u_{x,0}^{\text{out}} \partial_{x} T_{0}^{\text{out}} + u_{x,0}^{\text{out}} \partial_{x} T_{1}^{\text{out}} + u_{r,0}^{\text{out}} \partial_{r} T_{0}^{\text{out}} = \frac{1}{Pe} \left( \frac{1}{r} \partial_{r} \left( r \partial_{r} T_{0}^{\text{out}} \right) + \partial_{x} T_{0}^{\text{out}} + 2 \partial_{x} T_{1}^{\text{out}} + \partial_{\xi} T_{2}^{\text{out}} \right), \]  
(5.22c)

In principle \( T_{0}^{\text{out}} \) and \( T_{1}^{\text{out}} \) might depend on \( \xi \), however, in Section 5.5.1 we will show that the asymptotic momentum equations imply that \( T_{0}^{\text{out}} \) and \( T_{1}^{\text{out}} \) are only functions of the outer variables \( r \) and \( x \). This fact will allow us to obtain the homogenized equation in the outer region \( \Omega_{0} \). In fact, the homogenized equation will have the same form as the original equation (5.8c), but with the differences that it is now defined on a much simpler domain and that at the homogenized boundary \( \Gamma_{0} \) it satisfies a new boundary condition which takes into account the effects of wall-shape. This is the so-called effective boundary condition. In order to derive this effective boundary condition, we need to analyze the inner problem and use the method of matched asymptotic expansions to match the outer and the inner solutions.

5.4.2 Inner Equation

Now we turn our attention to the inner region. The first step we need to make is to rewrite the divergence \( \nabla \cdot u \) from (5.18) in a more convenient form to later expand the solution as an asymptotic series. To this extend we expand \( 1/(R_{w}^{0} + \epsilon \eta) \) in a geometric series as follows

\[ \frac{1}{R_{w}^{0} + \epsilon \eta} = \frac{1}{R_{w}^{0}} \left( 1 - \frac{\epsilon \eta}{R_{w}^{0}} + \left( \frac{\epsilon \eta}{R_{w}^{0}} \right)^{2} - \left( \frac{\epsilon \eta}{R_{w}^{0}} \right)^{3} \right) + \mathcal{O}(\epsilon^{4}). \]  
(5.23)

Substituting the previous expansion up to order \( \mathcal{O}(\epsilon^{2}) \) into (5.19), we obtain the following asymptotic expression for the divergence of a vector \( u \) in the inner region

\[ \nabla \cdot u = \frac{1}{\epsilon} \left( \partial_{r} u_{r}^{\text{in}} + \partial_{\xi} u_{\xi}^{\text{in}} \right) + \partial_{x} u_{x}^{\text{in}} + \frac{u_{r}^{\text{in}}}{R_{w}^{0}} - \frac{\epsilon \eta u_{r}^{\text{in}}}{(R_{w}^{0})^{2}} + \frac{\epsilon^{2} \eta^{2} u_{r}^{\text{in}}}{(R_{w}^{0})^{3}}. \]  
(5.24)

Now we can use (5.24) in combination with the expression for the gradient in (5.18), to rewrite the convective and diffusive terms of the temperature equation (5.8c) in the
inner region as follows

\[
\nabla \cdot (\nabla T^{\text{in}}) = \frac{1}{\epsilon^2} \partial_{\eta\eta} T^{\text{in}} + \frac{2}{\epsilon} \partial_{\xi \xi} T^{\text{in}} + \partial_{xx} T^{\text{in}} + \left( \frac{1}{\epsilon R_0^w} - \frac{\eta}{(R_0)^2} + \frac{\epsilon \eta^2}{(R_0^w)^2} \right) \partial_\eta T^{\text{in}},
\]

\[(5.25)\]

\[
\nabla \cdot (\nabla T^{\text{in}}) = \frac{1}{\epsilon} u^{\text{in}} \partial_{\eta} T^{\text{in}} + u^x \left( \partial_x T^{\text{in}} + \frac{1}{\epsilon} \partial_\xi T^{\text{in}} \right).
\]

\[(5.26)\]

Substituting the previous expressions into (5.8c), using the asymptotic ansatz for the inner region (5.15) and collecting terms of the same order in \(\epsilon\), we obtain the following equations

\[
\mathcal{O} \left( \frac{1}{\epsilon^2} \right) : \quad \partial_{\eta\eta} T^{\text{in}} + \partial_{\xi \xi} T^{\text{in}} = 0,
\]

\[(5.27a)\]

\[
\mathcal{O} \left( \frac{1}{\epsilon} \right) : \quad u^{\text{in}} \partial_{\eta} T^{\text{in}} + u^x \partial_x T^{\text{in}} = \frac{1}{P_0} \left( \partial_{\eta\eta} T^{\text{in}} + \partial_{\xi \xi} T^{\text{in}} + 2 \partial_\xi T^{\text{in}} + \frac{1}{R_0^w} \partial_\xi T^{\text{in}} \right).
\]

\[(5.27b)\]

It is interesting to notice that the form of equation (5.27a) coincides with the Laplacian operator in Cartesian coordinates, if we consider the inner variables \(\eta\) and \(\xi\) as a new coordinate system. This means that to leading order, the inner temperature does not feel the effects of curvature present in the cylindrical coordinate system; even though this might seem a bit strange at first. The result is consistent, because in the asymptotic limit \(\epsilon \to 0\), the wall \(\Gamma_\epsilon\) converges to the flat wall \(\Gamma_0\). Generally speaking, the inner region (which is of order \(\mathcal{O}(\epsilon)\)) only contains the effects of curvature in the term \(1/(R_0^w + \epsilon \eta)\), which we expanded in (5.23).

### 5.4.3 Boundary Conditions

Now we consider the asymptotic treatment of the boundary conditions (5.9a). By definition, the boundary \(\Gamma_\epsilon\) is always contained in the inner region. Therefore the inner solutions should satisfy (5.9a). The only difference that arises is that the inner solutions also depend on the fast variable \(\xi\). Hence we need to rewrite (5.9a) in terms of the inner variables.

The boundary condition which concerns the temperature equation is \(\mathbf{n} \cdot \nabla T = f_q\) at \(\Gamma_\epsilon\), where \(f_q\) is an \(\mathcal{O}(1)\) function. To transform the boundary condition, we need to recall
the expression for the normal vector $n$ from (2.50a). This reads

$$n = \frac{e_r - d_x R_w e_x}{\sqrt{1 + (d_x R_w)^2}} \quad (5.28)$$

where in our particular case the radius is given by

$$R_w(x) = R_w^0 + \epsilon R_{w_0}^\in(x). \quad (5.29)$$

Using the chain rule in (5.29) and the definition of $\xi$ in (5.10), we obtain the identity $d_x R_w = d_\xi R_{w_0}^\in$, which allows us to write $n$ as

$$n = \frac{e_r - d_\xi R_{w_0}^\in e_x}{\sqrt{1 + (d_\xi R_{w_0}^\in)^2}} \quad (5.30)$$

Combining the previous expression and the expression for $\nabla T$ from (5.18) we get

$$f_q = \frac{1}{\epsilon} \partial_\eta T^\in_0 - \partial_x T^\in d_\xi R_{w_0}^\in - \frac{1}{\epsilon} \partial_\xi T^\in d_\eta R_{w_0}^\in \frac{1}{\sqrt{1 + (d_\xi R_{w_0}^\in)^2}} \quad (5.31)$$

Substituting the asymptotic expansion (5.15) for $T^\in$, and using that (5.31) must hold for any value of $\epsilon$ (with $f_q \approx O(1)$). We obtain the following conditions

$$O\left(\frac{1}{\epsilon}\right) : \frac{\partial_\eta T^\in_0 - d_\xi R_{w_0}^\in \partial_\xi T^\in_0}{\sqrt{1 + (d_\xi R_{w_0}^\in)^2}} = 0, \quad (5.32a)$$

$$O(1) : \frac{\partial_\eta T^\in_1 - d_x T^\in_0 d_\xi R_{w_0}^\in - \partial_\xi T^\in_0 d_\eta R_{w_0}^\in}{\sqrt{1 + (d_\xi R_{w_0}^\in)^2}} = f_q. \quad (5.32b)$$

Expression (5.32a) is a homogeneous Neumann boundary condition for the local temperature field $T^\in_0$. We now have a microscopic system with respect to the local coordinate system $\eta$, $\xi$ and $x$.

### 5.4.4 Matching Conditions

The previous discussion provides us with a boundary condition at the boundary $\Gamma^\in$ of the inner domain $\Omega^\in$, which corresponds to the domain represented in terms of the local coordinates $\xi$ and $\eta$, as it is shown in Figure 5.2. The inner domain $\Omega^\in$ is a semi-infinite strip which "contains" the corrugations of the pipe. In the local coordinate system the
5.4 Homogenization of the Temperature Equation

corrugation is denoted by $\Gamma_{in}$. To be more precise, we define the inner domain

$$
\Omega_{in} := \{ \xi e_1 + \eta e_2 \in \mathbb{R}^2 \mid 0 \leq \xi \leq L_{in}, -\infty \leq \eta \leq R_{in}(\xi) \},
$$

(5.33)

where $e_1$ and $e_2$ are the canonical Cartesian basis vectors, and $L_{in}$ is the period of the function $R_{in}$.

The inner solution by definition, needs to be valid in the inner region $\Omega_{in}$. Due to this, the leading order inner solution $T_{in}^0$ needs to satisfy (5.27a) in $\Omega_{in}$ and the boundary condition (5.32a) at $\Gamma_{in}$. In an analogous way $T_{in}^1$ satisfies (5.27b) in $\Omega_{in}$ and the boundary condition (5.32b) at $\Gamma_{in}$. Following our asymptotic ansatz (5.12), (5.13), all functions are periodic with respect to the fast variable $\xi$. Hence, we impose periodic boundary conditions at the left and right boundaries $\Gamma_l$ and $\Gamma_r$, respectively.

The boundary condition when $\eta \to -\infty$ has not been yet discussed. This boundary condition will be the one responsible for bringing information related to the small scale geometry in $\Omega_{in}$, to the large scale behavior in the domain $\Omega_0$. In the field of homogenization, the problem to be solved in $\Omega_{in}$ is referred to as the cell problem, and such cell problems are an essential part in the upscaling of singularly perturbed problems [60].

We now face the problem of how to define the boundary condition of the inner solution at $\eta \to -\infty$. In the method of matched asymptotic expansions, a uniformly valid solution is obtained by matching the outer and the inner expansions according to some matching principle. The general matching principle states that the outer solution and the inner expansions represent the same function in an intermediate region of overlap. This overlap region may be described by a stretched variable asymptotically in between the outer and the inner scales (between $O(1)$ and $O(\epsilon)$ for our problem) [64]. A simpler procedure than using the intermediate stretching variable, is the Van Dyke’s matching
rule. This rule states that the outer expansion, when expressed in terms of the inner variables and expanded for small $\epsilon$, must agree with the inner solution, when expressed in terms of the outer variables and expanded for small $\epsilon$ [132]. There are also other matching principles for high order due to Shivamoggi, which are obtained by writing a Taylor expansion of the asymptotic series of the outer solution near the boundary layer, and by rewriting this expansion in terms of the boundary layer variable [106].

In our particular case, however, it is enough to impose limit matching rules. The basic idea of such a matching rule is that the asymptotic behavior of the outer solution when extended into the inner region must be the same as that of the inner solution when it is extended into the outer region [132]. To have a better picture of this, it is convenient to recall the definition of our boundary layer variable

$$
\eta = \frac{r - R_0^w}{\epsilon}.
$$

(5.34)

This variable is used in order to be able to expand the solution in an asymptotic series around $r = R_0^w$. When $\eta$ is fixed and $\epsilon \to 0$, then $r \to R_0^w$, while if $r < R_0^w$ is fixed and $\epsilon \to 0$, then $\eta \to -\infty$. This fact also explains why the inner domain $\Omega^\text{in}$ in terms of the local variables is a semi-infinite strip.

The reasoning behind limit matching rules is the following. Since both expansions $T^\text{in}$ and $T^\text{out}$ represent the solution to the same problem, the value of the outer solution as one goes into the inner region ($r \to R_0^w$), and the value of the inner solution as one goes into the outer region ($\eta \to -\infty$) should coincide. This is in fact the original matching principle due to Prandtl [93]

$$
T^\text{out} \left( R_0^w, x, \xi; \epsilon \right) = \lim_{\eta \to -\infty} T^\text{in} \left( \eta, x, \xi; \epsilon \right).
$$

(5.35)

After expanding $T^\text{in}$ and $T^\text{out}$ in asymptotic series with respect to $\epsilon$, the matching principles takes the form

$$
T^\text{out}_i \left( R_0^w, x, \xi \right) = \lim_{\eta \to -\infty} T^\text{in}_i \left( \eta, x, \xi \right), \quad \text{for } i = 0, 1, 2, \ldots
$$

(5.36)

Since our problem at hand (5.8) is equipped with the non-homogenous Neumann boundary condition (5.6), we can expect the homogenized boundary condition to involve derivatives. In fact, in addition to the Prandtl matching condition (5.35), we will also need a matching condition between the derivatives of the outer and the inner solutions. To obtain this matching condition we recall that in the outer region $\Omega_0$ we have $\partial_r T = \partial_r T^\text{out}$, while in the inner region $\Omega^\text{in}$ we have $\partial_r T = \frac{4}{\epsilon} \partial_\eta T^\text{in}$. Assuming that the solution is continuously differentiable, then the derivative in the outer region as one goes into the inner region should match the derivative in the inner region as one goes
into the outer region. In analogy to the matching condition (5.35), we have the following matching condition for the derivatives
\[
\partial_r T_{\text{out}}^0 \left(R_0, x, \xi; \epsilon\right) = \lim_{\eta \to -\infty} \frac{1}{\epsilon} \partial_\eta T_{\text{in}}^0 \left(\eta, x, \xi; \epsilon\right).
\] (5.37)

By substituting the asymptotic expansions into (5.37) and grouping terms with respect to their order in \(\epsilon\) we obtain the conditions
\[
\lim_{\eta \to -\infty} \partial_\eta T_{\text{in}}^0 \left(\eta, x, \xi; \epsilon\right) = 0, \quad (5.38a)
\]
\[
\lim_{\eta \to -\infty} \partial_\eta T_{\text{in}}^1 \left(\eta, x, \xi; \epsilon\right) = \partial_r T_{\text{out}}^0 \left(R_0, x, \xi\right).
\] (5.38b)

The previous matching conditions for the derivatives are crucial for transferring the local information from the inner solution into the outer solution. This transfer of information takes place through the homogenized boundary \(\Gamma_0\). We will now derive this boundary condition which we will refer to as the effective boundary condition.

### 5.4.5 Effective Boundary Condition

Now we proceed to use the equations and boundary conditions which we have obtained in order to continue the homogenization procedure. We start with the equations which we obtained for the outer solution. From (5.22a) we have \(\partial_{\xi x} T_{\text{out}}^0 = 0\) and hence \(T_{\text{out}}^0\) can at most be a linear function with respect to the fast variable \(\xi\). However, according to our asymptotic ansatz (5.12), \(T_{\text{out}}^0\) needs to be periodic in \(\xi\), and therefore we conclude that \(T_{\text{out}}^0\) is independent of \(\xi\). This means that \(T_{\text{out}}^0\) can only be function of the outer variables \(x\) and \(r\), i.e., \(T_{\text{out}}^0 = T_{\text{out}}^0(r, x)\).

The previous result in particular implies \(\partial_{\xi x} T_{\text{out}}^0 \equiv 0\). Using this information we can simplify (5.22b) and obtain
\[
\partial_{\xi x} T_{\text{out}}^1 = 0.
\] (5.39)

Using exactly the same argument as for \(T_{\text{out}}^0\), we can conclude that \(T_{\text{out}}^1 = T_{\text{out}}^1(r, x)\). This in turn implies \(\partial_{\xi x} T_{\text{out}}^1 \equiv 0\). With these intermediate results, we can eliminate some terms from equation (5.22c), and obtain the following equation
\[
u_{r,0} \partial_x T_{\text{out}}^0 + u_{r,0} \partial_r T_{\text{out}}^0 = \frac{1}{\rho c} \left(1 + \frac{1}{r} \partial_r \left(r \partial_x T_{\text{out}}^0 \right) + \partial_{xx} T_{\text{out}}^0 + \partial_{\xi x} T_{\text{out}}^2\right). \quad (5.40)
\]

Now we can integrate the previous equation with respect to \(\xi\) over one period, i.e., \(0 \leq \xi \leq L\). Due to the periodicity in \(\xi\), the integral of the term \(\partial_{\xi x} T_{\text{out}}^2\) vanishes,
leaving us with the equation
\[
\int_0^{L_{in}} u_{x,0}^\text{out} \partial_x T_0^\text{out} + u_{r,0}^\text{out} \partial_r T_0^\text{out} \, d\xi = \frac{1}{Pe} L_{in} \left( \frac{1}{r} \partial_r \left( r \partial_r T_0^\text{out} \right) + \partial_{xx} T_0^\text{out} \right).
\] (5.41)

The left hand side term in the previous equation seems to be of a complicated nature, however, in Section 5.5.1 we show that both \(u_{x,0}^\text{out}\) and \(u_{r,0}^\text{out}\) are also independent of \(\xi\). This comes as a result of the asymptotic expansions of the momentum equations (see (5.61a) and (5.64a)). Using the fact that \(u_{x,0}^\text{out}\) and \(u_{r,0}^\text{out}\) are independent of \(\xi\), the integration just adds the trivial multiplicative constant \(L_{in}\). Hence after dividing by \(L_{in}\), we obtain the homogenized equation which holds in the domain \(\Omega_0\).

\[
u_{x,0}^\text{out} \partial_x T_0^\text{out} + u_{r,0}^\text{out} \partial_r T_0^\text{out} = \frac{1}{Pe} \left( \frac{1}{r} \partial_r \left( r \partial_r T_0^\text{out} \right) + \partial_{xx} T_0^\text{out} \right).\] (5.42)

We notice that (5.42) has exactly the same form as the original convection diffusion equation (5.8c). Since \(\Omega_0\) contains \(\Gamma_c\), \(\Gamma_{\text{bot}}\) and \(\Gamma_{\text{top}}\), the conditions at these boundaries are inherited from the original problem. For example, \(\partial_r u_{x,0}^\text{out} = 0\) on \(\Gamma_c\), just as in the boundary condition for the original problem (5.9a). This means that the outer solution only can obtain information from the small scale geometry through the boundary condition at the homogenized boundary \(\Gamma_0\).

The boundary condition at the homogenized boundary \(\Gamma_0\) requires special treatment, and in fact, deriving this effective boundary condition can be seen as the main goal of the formal homogenization procedure. In order to obtain this effective boundary condition, we need to make use of the inner solution (cell problem) in \(\Omega_{in}\), and of the matching conditions (5.35) and (5.38).

We consider first equation (5.27a). This equation is the traditional Laplace equation in the local Cartesian coordinates \(\xi\) and \(\eta\). The boundary condition at \(\Gamma_{in}\) for this equation is (5.32a). This condition can be written as a homogeneous Neumann boundary condition by noting that the outer unit normal vector to the boundary \(\Gamma_{in}\) can be written as

\[
n_{in} = \frac{\mathbf{e}_2 - d_\xi R_{in}^w \mathbf{e}_1}{\sqrt{1 + \left( d_\xi R_{in}^w \right)^2}}\] (5.43)

This allows us to write (5.32a) as \(n_{in} \cdot \nabla_{\xi,\eta} T_{in}^0 = 0\), where \(\nabla_{\xi,\eta}\) denotes the gradient in the local Cartesian coordinate system. In addition, we have the matching condition (5.36) which describes the behavior of \(T_{0}^\text{in}\) when \(\eta \to -\infty\). Altogether we have that \(T_{0}^\text{in}\) needs
5.4 Homogenization of the Temperature Equation

to satisfy
\[
\begin{align*}
\partial_{\xi\xi} T_0^\text{in} + \partial_{\eta\eta} T_0^\text{in} &= 0, & \text{in } \Omega^\text{in} \quad (5.44a) \\
\mathbf{n}^\text{in} \cdot \nabla_{\xi,\eta} T_0^\text{in} &= 0, & \text{at } \Gamma^\text{in} \quad (5.44b) \\
\lim_{\eta \to -\infty} T_0^\text{in}(\eta, x, \xi) &= T_0^\text{out}(R_0^w, x), & \quad (5.44c) \\
T_0^\text{in}(\eta, x, 0) &= T_0^\text{in}(\eta, x, L^\text{in}), & \quad (5.44d)
\end{align*}
\]

where the last condition is the periodicity with respect to the fast variable \(\xi\), which is part of our asymptotic ansatz for the multiple scales.

It is not difficult to see that \(T_0^\text{in}(\eta, x, \xi) = T_0^\text{out}(R_0^w, x)\) is solution to (5.44). Since \(T_0^\text{in}(\eta, x, \xi) = T_0^\text{out}(x, R_0^w)\), in particular we have that \(\partial_{\eta} T_0^\text{in} = \partial_{\xi} T_0^\text{in} = 0\). This allows us to simplify (5.27b) and we obtain another Laplace equation in Cartesian coordinates for \(T_1^\text{in}\). This equation is provided with the boundary condition (5.32b), and the matching condition (5.38). After the above mentioned simplification, (5.27b) takes the form
\[
\partial_{\eta\eta} T_1^\text{in} + \partial_{\xi\xi} T_1^\text{in} = 0.
\]

We rewrite this equation in divergence form as
\[
\nabla_{\xi,\eta} \cdot \left( \nabla_{\xi,\eta} T_1^\text{in} \right) = 0.
\]

Now we will integrate this equation over the domain \(\Omega^\text{in}\). In order to use the divergence theorem, we write
\[
\Omega^\text{in} = \lim_{M \to -\infty} \Omega^M,
\]

where
\[
\Omega^M := \left\{ \xi \mathbf{e}_1 + M \mathbf{e}_2 \in \mathbb{R}^2 \mid 0 \leq \xi \leq L^\text{in}, \ M \leq \eta \leq R_0^w(\xi) \right\}.
\]

Integrating (5.46) over \(\Omega^M\) and applying the divergence theorem we obtain
\[
\int_{\Gamma^M} \mathbf{n}^\text{in} \cdot \nabla_{\xi,\eta} T_1^\text{in} \, dl - \int_{\Gamma_{\xi} \cap \Omega^M} \mathbf{e}_1 \cdot \nabla_{\xi,\eta} T_1^\text{in} \, dl + \int_{\Gamma_{\eta} \cap \Omega^M} \mathbf{e}_2 \cdot \nabla_{\xi,\eta} T_1^\text{in} \, dl = 0,
\]

where \(dl\) represents the length element and \(\Gamma^M := \{ \xi \mathbf{e}_1 + M \mathbf{e}_2 \in \mathbb{R}^2 \mid 0 \leq \xi \leq L^\text{in} \}\) is the lower boundary of \(\Omega^M\).

Due to the periodicity in \(\xi\), the second and third integral in (5.49) cancel. At the bound-
ary $\Gamma^{in}$, we can substitute the boundary condition (5.32b) yielding
\[
\int_{\Gamma^{in}} \partial_\eta T^in_1 \, dl = \int_{\Gamma^{in}} f_q + \frac{\partial_x T^in_0 \, d_x R^in_w}{\sqrt{1 + \left( d_x R^in_w \right)^2}} \, dl.
\] (5.50)

Now we parameterize the boundaries $\Gamma^M$ and $\Gamma^{in}$ in terms of $0 \leq \xi \leq L^in$ as $\xi e_1 + M e_2$ and $\xi e_1 + R^in_w(\xi) e_2$, respectively. Using these parameterizations, the previous equation becomes
\[
\int_0^{L^in} \partial_\eta T^in_1 (M, x, \xi) \, d\xi = f_q \int_0^{L^in} \sqrt{1 + \left( d_x R^in_w(\xi) \right)^2} \, d\xi + \partial_x T^in_0 \int_0^{L^in} \left( d_x R^in_w(\xi) \right) \, d\xi,
\] (5.51)
where we have used the fact that $T^in_0$ and $f_q$ are independent of $\xi$. Since $R^in_w$ is a periodic function, the last integral is equal to 0. We can rewrite the integral on the left by taking the limit $M \to -\infty$ and using the matching condition for the radial derivative (5.38b). We obtain
\[
\lim_{M \to -\infty} \int_0^{L^in} \partial_\eta T^in_1 (M, x, \xi) \, d\xi = \int_0^{L^in} \partial_\eta T^{out} \left( R^0_w, x \right) \, d\xi = L^in \partial_\eta T^{out}_0 (r, x).
\] (5.52)

Finally substituting this expression in (5.51) we obtain the effective boundary condition
\[
\partial_\eta T^{out}_0 = \frac{f_q}{L^in} \int_0^{L^in} \sqrt{1 + \left( d_x R^in_w \right)^2} \, d\xi \quad \text{at } \Gamma_0.
\] (5.53)

The constant which multiplies $f_q$ can be interpreted as a correction factor which modifies the boundary condition in order to take into account the effects of the corrugations into the homogenized solution. This factor is given by the ratio of the lengths of the curves $\Gamma^{in}$ and $\Gamma^M$.

Summarizing, we have transformed the problem of solving the temperature equation (5.8c) in the corrugated domain $\Omega_\epsilon$ (with multiple scales) with boundary conditions (5.9a), into the problem of solving the homogenized temperature equation (5.42) in the much simpler homogenized domain $\Omega_0$, with the effective boundary condition (5.53). In a later section we will show the computational benefits of using this upscaled equations and effective boundary condition. Now we will discuss the homogenization of the continuity and momentum equations equations (5.8a) and (5.8b).
5.5 Homogenization of the Flow Equations

Now we proceed to apply formal homogenization to equations (5.8a) and (5.8b). We will make use of the differential operators defined in (5.16) and we also need to use the following expressions for the vector Laplacian and the material derivative for an axially symmetric flow

\[
\nabla^2 \mathbf{u} = \left( \frac{1}{r} \partial_r (r \partial_r u_r) + \partial_{xx} u_r - \frac{1}{r^2} u_r \right) \mathbf{e}_r + \left( \frac{1}{r} \partial_r (r \partial_r u_x) + \partial_{xx} u_x \right) \mathbf{e}_x, \tag{5.54}
\]

\[
\mathbf{u} \cdot \nabla \mathbf{u} = (u_r \partial_r u_r + u_x \partial_x u_r) \mathbf{e}_r + (u_r \partial_r u_x + u_x \partial_x u_x) \mathbf{e}_x. \tag{5.55}
\]

5.5.1 Outer Solution

We recall the differentiation rules for the outer solutions

\[
\partial_r p = \partial_r p^{\text{out}}, \quad \partial_r r p^{\text{out}}, \quad \partial_x p = \partial_x p^{\text{out}} + \frac{1}{\epsilon} \partial_\xi p^{\text{out}}, \quad \partial_{xx} p = \partial_{xx} p^{\text{out}} + \frac{2}{\epsilon} \partial_\xi p^{\text{out}} + \frac{1}{\epsilon^2} \partial_{\xi \xi} p^{\text{out}}. \tag{5.56}
\]

The same rules hold for any other scalar field in the outer region, such as the components of the velocity \( \mathbf{u} \). We start by writing the continuity equation (5.8a) in the outer region, this yields

\[
\frac{1}{r} \partial_r \left( r u^{\text{out}}_r \right) + \partial_x u^{\text{out}}_x + \frac{1}{\epsilon} \partial_\xi u^{\text{out}}_x = 0, \tag{5.57}
\]

expanding the outer solutions according to our asymptotic ansatz (5.14) and collecting term in orders of \( \epsilon \) we obtain the equations

\[
\mathcal{O} \left( \frac{1}{\epsilon} \right): \quad \partial_\xi u^{\text{out}}_{x,0} = 0, \tag{5.58a}
\]

\[
\mathcal{O}(1): \quad \frac{1}{r} \partial_r \left( r u^{\text{out}}_{r,0} \right) + \partial_x u^{\text{out}}_{r,0} + \partial_\xi u^{\text{out}}_{r,1} = 0. \tag{5.58b}
\]

Now we will consider the radial and axial components of equation (5.8b). First we use (5.54) to obtain the equations for the radial and axial momentum, i.e.,

Radial: \( u_r \partial_r u_r + u_x \partial_x u_r = -\partial_r p + \frac{1}{\text{Re}} \left( \frac{1}{r} \partial_r (r \partial_r u_r) + \partial_{xx} u_r - \frac{1}{r^2} u_r \right) \), \tag{5.59a}

Axial: \( u_r \partial_r u_x + u_x \partial_x u_x = -\partial_x p + \frac{1}{\text{Re}} \left( \frac{1}{r} \partial_r (r \partial_r u_x) + \partial_{xx} u_x \right) + T. \) \tag{5.59b}
Using the differentiation rules (5.56), the equation of radial momentum (5.59a) takes the form

\[
u_r^{\text{out}} \partial_r u_r^{\text{out}} + u_x^{\text{out}} \left( \partial_x u_r^{\text{out}} + \frac{1}{\epsilon} \partial_x u_x^{\text{out}} \right) = -\partial_r p^{\text{out}} + \frac{1}{\text{Re}} \left( \frac{1}{r} \partial_r \left( r \partial_r u_r^{\text{out}} \right) - \partial_x u_r^{\text{out}} \right) + \frac{2}{\epsilon} \partial_x u_x^{\text{out}} + \frac{1}{\epsilon^2} \partial_{\xi\xi} u_r^{\text{out}} - \frac{1}{r^2} u_r^{\text{out}} \right) . \tag{5.60}
\]

After substituting the asymptotic expansion ansatz and collecting terms with respect to their order in \( \epsilon \), we obtain the following equations

\[
O \left( \frac{1}{\epsilon^2} \right) : \quad \partial_{\xi\xi} u_r^{\text{out},0} = 0 , \tag{5.61a}
\]

\[
O \left( \frac{1}{\epsilon} \right) : \quad u_r^{\text{out},0} \partial_{x,0} u_r^{\text{out}} = \frac{1}{\text{Re}} \left( 2 \partial_x u_r^{\text{out},0} + \partial_x u_r^{\text{out},1} \right) , \tag{5.61b}
\]

\[
O(1) : \quad u_r^{\text{out},0} \partial_{x,0} u_r^{\text{out}} + u_x^{\text{out},0} \partial_x u_r^{\text{out}} + u_{x,1} \partial_{x,0} u_r^{\text{out}} + u_{x,0} \partial_x u_{x,0} = -\partial_x p^{\text{out}} + \frac{1}{r} \partial_r \left( r \partial_r u_r^{\text{out}} \right) + \partial_x u_r^{\text{out},0} - \frac{2}{r^2} u_r^{\text{out},0} + 2 \partial_x u_r^{\text{out},1} + \partial_{\xi\xi} u_r^{\text{out},2} \right) . \tag{5.61c}
\]

We continue now with the axial component of the momentum equation. After using the differentiation rules (5.56) in the outer region for the axial momentum equation (5.59b), we obtain the following equation

\[
u_x^{\text{out}} \partial_r u_x^{\text{out}} + u_x^{\text{out}} \left( \partial_x u_x^{\text{out}} + \frac{1}{\epsilon} \partial_x u_x^{\text{out}} \right) = -\partial_r p^{\text{out}} - \frac{1}{\epsilon} \partial_x p^{\text{out}} + \frac{1}{\text{Re}} \left( \frac{1}{r} \partial_r \left( r \partial_r u_x^{\text{out}} \right) + \partial_x u_x^{\text{out},0} + \frac{2}{\epsilon} \partial_x u_x^{\text{out},1} + \frac{1}{\epsilon^2} \partial_{\xi\xi} u_x^{\text{out}} \right) + T^{\text{out}} . \tag{5.62}
\]

By expanding in asymptotic series and grouping terms with respect to their order in \( \epsilon \), we obtain

\[
O \left( \frac{1}{\epsilon^2} \right) : \quad \partial_{\xi\xi} u_x^{\text{out},0} = 0 , \tag{5.64a}
\]

\[
O \left( \frac{1}{\epsilon} \right) : \quad u_x^{\text{out},0} \partial_{x,0} u_x^{\text{out}} = -\partial_{x,0} p^{\text{out}} + \frac{1}{\text{Re}} \left( 2 \partial_x u_x^{\text{out},0} + \partial_x u_x^{\text{out},1} \right) , \tag{5.64b}
\]

\[
O(1) : \quad u_r^{\text{out},0} \partial_{x,0} u_r^{\text{out}} + u_{x,0} \partial_{x,0} u_x^{\text{out}} + u_{x,0} \partial_{x,1} u_x^{\text{out}} + u_{x,1} \partial_{x,0} u_x^{\text{out}} = -\partial_r p^{\text{out}} - \partial_{x,0} p^{\text{out}} + \frac{1}{r} \partial_r \left( r \partial_r u_r^{\text{out},0} \right) + \partial_x u_x^{\text{out},0} + 2 \partial_x u_x^{\text{out},1} + \partial_{\xi\xi} u_x^{\text{out},2} \right) + T^{\text{out}} . \tag{5.64c}
\]
5.5 Homogenization of the Flow Equations

5.5.2 Homogenized Equations

Now we will use the obtained equations to derive the homogenized flow equations in the domain \( \Omega_0 \). Since the type of arguments are of the same nature as the ones we used to derive the homogenized the temperature equations, we present the derivations more directly. We start with the continuity equation.

On the one hand, from (5.58a), we conclude that \( u_{x,0}^{\text{out}} = u_{x,0}^{\text{out}}(r, x) \). On the other hand, the periodicity with respect to \( \xi \) together with equation (5.61a), imply \( u_{r,0}^{\text{out}} = u_{r,0}^{\text{out}}(r, x) \). Now we know that both velocity component \( u_{r,0}^{\text{out}} \) and \( u_{x,0}^{\text{out}} \) are independent of \( \xi \). Therefore we can now integrate (5.58b) on \( 0 \leq \xi \leq L^\text{in} \). Using the periodicity in \( \xi \), the integral of the term \( \partial_\xi u_{x,1}^{\text{out}} \) vanishes leaving us with the integrals of terms which are independent of \( \xi \). Therefore the integration becomes trivial, and we obtain the following homogenized continuity equation which holds in the domain \( \Omega_0 \)

\[
\frac{1}{r} \frac{\partial}{\partial r} \left( ru_{r,0}^{\text{out}} \right) + \partial_x u_{x,0}^{\text{out}} = 0.
\] (5.65)

We continue with the radial momentum equation. We already now that \( u_{r,0}^{\text{out}} = u_{r,0}^{\text{out}}(r, x) \). This in turn implies \( \partial_\xi u_{r,0}^{\text{out}} = 0 \), and hence we can eliminate these terms from equation (5.61b), which then simplifies to \( \partial_\xi u_{r,1}^{\text{out}} = 0 \). Again due to periodicity, this relation implies \( u_{r,1}^{\text{out}} = u_{r,1}^{\text{out}}(r, x) \), which then allows us to eliminate all the terms involving \( u_{r,1}^{\text{out}} \) from (5.61c). Finally, we can integrate the resulting equation with respect to \( \xi \) from 0 to \( L^\text{in} \). The integral of the term \( \partial_\xi u_{r,2}^{\text{out}} \) becomes 0 due to periodicity. All the other functions are independent of \( \xi \) and hence the integration with respect to \( \xi \) ends in a trivial multiplication by the constant \( L^\text{in} \), which we can simply cancel out. Altogether we arrive at the following equation for the radial momentum in the homogenized domain \( \Omega_0 \).

\[
u_{r,0} \partial_r u_{r,0}^{\text{out}} + u_{x,0}^{\text{out}} \partial_x u_{r,0}^{\text{out}} = -\partial_r p_0^{\text{out}} +\frac{1}{\text{Re}} \left( \frac{1}{r} \partial_r \left( ru_{r,0}^{\text{out}} \right) \right) \left( \partial_x u_{x,0}^{\text{out}} - \frac{1}{r^2} u_{r,0}^{\text{out}} \right). \] (5.66)

Finally we consider the axial momentum equation. We already know that \( u_{x,0}^{\text{out}} \) is independent of \( \xi \), i.e., \( u_{x,0}^{\text{out}} = u_{x,0}^{\text{out}}(r, x) \). With this information (5.64b) simplifies to \( \partial_\xi p_0^{\text{out}} = \partial_\xi u_{x,1}^{\text{out}} / \text{Re} \). To proceed further we need to use the continuity equation. First from (5.61a) we conclude that \( u_{r,0}^{\text{out}} \equiv 0 \). Then, differentiating (5.58b) with respect to \( \xi \) we obtain \( \partial_\xi u_{x,1}^{\text{out}} = 0 \) and therefore we find out \( u_{x,1}^{\text{out}} = u_{x,1}^{\text{out}}(r, x) \). This allows us to simplify (5.64b) further and we obtain \( \partial_\xi p_0^{\text{out}} = 0 \). Therefore we conclude that also \( p_0^{\text{out}} \) is independent of \( \xi \), i.e., \( p_0^{\text{out}} = p_0^{\text{out}}(r, x) \). We can now use this information to simplify (5.64c).

In addition we will also integrate the equation with respect to \( \xi \) over \( 0 \leq \xi \leq L^\text{in} \). The integrals of \( -\partial_\xi p_1^{\text{out}} \) and \( \partial_\xi u_{x,2}^{\text{out}} \) vanish due to periodicity. After dividing by \( L^\text{in} \), we
obtain the following homogenized axial momentum equation in the domain \( \Omega_0 \)

\[
\begin{align*}
    u_{r,0}^{\text{out}} \partial_r u_{x,0}^{\text{out}} + u_{x,0}^{\text{out}} \partial_x u_{x,0}^{\text{out}} &= -\partial_x p_0^{\text{out}} + \frac{1}{Re} \left( \frac{1}{r} \partial_r \left( r \partial_r u_{x,0}^{\text{out}} \right) + \partial_x u_{x,0}^{\text{out}} \right) + T_0^{\text{out}}. \quad (5.67)
\end{align*}
\]

We need to provide the homogenized equations (5.65), (5.66), and (5.67), with the effective boundary condition at the homogenized boundary \( \Gamma_0 \). In order to do this, we need to recur to the inner solution.

### 5.5.3 Inner Solution

We now consider the solution of the fluid equations in the inner region. In the following derivations we will make use of the rules of differentiation with respect to \( \xi \) in (5.56) and also of the following differentiation rules with respect to \( \eta \)

\[
\begin{align*}
    \frac{1}{r} \partial_r \left( ru_r \right) &= \frac{1}{\epsilon} \partial_\eta u_r + \frac{1}{R_0^w + \epsilon \eta} u_r \\
    &= \frac{1}{\epsilon} \partial_\eta u_r + \frac{1}{R_0^w} \left( 1 - \frac{\epsilon \eta}{R_0^w} + \frac{(\epsilon \eta)^2}{(R_0^w)^2} - \frac{(\epsilon \eta)^3}{(R_0^w)^3} \right) u_r + \mathcal{O} \left( \epsilon^4 \right), \quad (5.69)
\end{align*}
\]

the last equality is obtained after developing the fraction in a Taylor series (see (5.23)).

Using these rules and our asymptotic ansatz, we can rewrite the continuity equation (5.8a) in the inner region and obtain

\[
\begin{align*}
    \frac{1}{\epsilon} \partial_\eta u_r + \frac{u_r}{R_0^w} - \frac{\epsilon \eta u_r}{(R_0^w)^2} + \partial_x u_x + \frac{1}{\epsilon} \partial_\xi u_x + \mathcal{O} \left( \epsilon^2 \right) &= 0. \quad (5.70)
\end{align*}
\]

Following the usual procedure, we can expand in asymptotic series and collect the terms with respect to their order in \( \epsilon \). In this particular case we will need only the leading order equation, i.e.,

\[
\mathcal{O} \left( \frac{1}{\epsilon} \right) : \quad \partial_\eta u_{r,0}^{\text{in}} + \partial_\xi u_{x,0}^{\text{in}} = 0. \quad (5.71)
\]

In the case of the radial momentum equation, we have to apply the differentiation rules (5.68) and the following geometric series

\[
\frac{1}{R_0^w + \epsilon \eta} = \frac{1}{R_0^w} - \frac{2 \epsilon \eta}{(R_0^w)^2} + \mathcal{O} \left( \epsilon^2 \right). \quad (5.72)
\]
After applying these identities, the radial equation (5.59a) in the inner region takes the form

\[
\frac{1}{\epsilon} u_{r}^{in} \partial_{\eta} u_{r}^{in} + u_{x}^{in} \partial_{x} u_{r}^{in} + \frac{1}{\epsilon} u_{x}^{in} \partial_{\xi} u_{r}^{in} = -\frac{1}{\epsilon} \partial_{\eta} p^{in} + \frac{1}{\epsilon} \partial_{\eta} u_{x}^{in} \partial_{\eta} u_{x}^{in} + \frac{1}{\epsilon} \partial_{x} u_{r}^{in} + \frac{1}{\epsilon} \partial_{x} u_{x}^{in} - \frac{u_{r}^{in}}{R_{w}^{0}} + \mathcal{O}(\epsilon). \tag{5.73}
\]

The equation which we obtain from the leading order terms is

\[
\mathcal{O}\left(\frac{1}{\epsilon^2}\right): \quad \partial_{\eta} u_{r,0}^{in} + \partial_{\xi} u_{r,0}^{in} = 0. \tag{5.75}
\]

In the case of the axial momentum equation (5.59b), we can also use the differentiation rules and rewrite this equation as

\[
\frac{1}{\epsilon} u_{r}^{in} \partial_{\eta} u_{x}^{in} + u_{x}^{in} \partial_{x} u_{x}^{in} + \frac{1}{\epsilon} u_{x}^{in} \partial_{\xi} u_{x}^{in} = -\partial_{x} p^{in} - \frac{1}{\epsilon} \partial_{\xi} p^{in} + \frac{1}{\epsilon} \partial_{\eta} u_{x}^{in} \partial_{\eta} u_{x}^{in} + \frac{1}{\epsilon} \partial_{x} u_{x}^{in} + \frac{1}{\epsilon} \partial_{x} u_{x}^{in} - \frac{u_{x}^{in}}{R_{w}^{0}} + \mathcal{O}(\epsilon).
\]

The equation which we obtain from the leading order terms is

\[
\mathcal{O}\left(\frac{1}{\epsilon^2}\right): \quad \partial_{\eta} u_{x,0}^{in} + \partial_{\xi} u_{x,0}^{in} = 0. \tag{5.77}
\]

### 5.5.4 Effective Boundary Condition

In order to obtain the effective boundary condition at \(\Gamma_0\) for the fluid flow equations, we will only need the traditional matching principle due to Prandtl; see (5.35) \([93]\), but now applied to the velocity components, i.e.,

\[
\lim_{r \to R_{w}^{0}} u_{r}^{out}(r, x, \xi) = \lim_{\eta \to -\infty} u_{r}^{in}(\eta, x, \xi), \quad \lim_{r \to R_{w}^{0}} u_{x}^{out}(r, x, \xi) = \lim_{\eta \to -\infty} u_{x}^{in}(\eta, x, \xi). \tag{5.78}
\]

Thus we need to determine the inner solution first. Since the inner region is close to the wall of the pipe, the boundary condition at \(\Gamma_{in}\) is inherited from the boundary condition at \(\Gamma\), i.e., \(u^{in} = 0\) at \(\Gamma^{in}\). In particular we have \(u_{r,0}^{in} = 0\) and \(u_{x,0}^{in} = 0\) at \(\Gamma^{in}\), see Figure 5.2. Consequently, the solution to the Laplace equations (5.75) and (5.77) are \(u_{r,0}^{in} \equiv 0\).
Homogenization of the Boussinesq Equations

and $u_{r,0}^{in} \equiv 0$. In contrast with the case of the Neumann boundary condition for the temperature equation, we can straightforwardly apply the matching principle (5.78) and obtain the effective boundary condition, which turns out to be the no-slip boundary condition at the homogenized boundary $\Gamma_0$, namely

$$u_{r,0}^{out} = 0, \quad u_{x,0}^{out} = 0, \quad \text{at} \quad \Gamma_0. \quad (5.79)$$

5.6 Homogenized Boussinesq System of Equations

The previous derivation provides us with the boundary condition required to close the homogenized flow equations (5.8a), (5.67) and (5.59a). Also in Section 5.4 we obtained the homogenized temperature equation (5.42), and its respective effective boundary condition (5.31). The results of our formal asymptotics are summarized in the following homogenized system of equations which holds in the homogenized domain $\Omega_0$, i.e.,

$$\frac{1}{r} \partial_r \left( r u_{r,0}^{out} \right) + \partial_x u_{x,0}^{out} = 0, \quad (5.80a)$$

$$u_{r,0}^{out} \partial_r u_{r,0}^{out} + u_{x,0}^{out} \partial_x u_{x,0}^{out} = -\partial_r p_0^{out} + \frac{1}{Re} \left( \frac{1}{r} \partial_r \left( r \partial_r u_{r,0}^{out} \right) + \partial_x u_{x,0}^{out} \right) - \frac{1}{r} u_{r,0}^{out}, \quad (5.80b)$$

$$u_{r,0}^{out} \partial_r T_0^{out} + u_{x,0}^{out} \partial_x T_0^{out} = \frac{1}{Pe} \left( \frac{1}{r} \partial_r \left( r \partial_r T_0^{out} \right) + \partial_x T_0^{out} \right), \quad (5.80c)$$

subject to the effective boundary conditions

$$u_{r,0}^{out} = 0, \quad u_{x,0}^{out} = 0, \quad \partial_r T_0^{out} = \frac{f q}{L_{in}} \int_0^{L_{in}} \sqrt{1 + \left( d_\xi R_{in}^2 \right)^2} d_\xi, \quad \text{at} \quad \Gamma_0. \quad (5.81)$$

The great advantage is that now our domain does not include the multiple scales. Multiple scales can make the direct numerical solution very complicated and sometimes too expensive to be solved in practice. The homogenized model avoids this difficulty, but at the same time the effects of the wall-shape are taken into account by the effective boundary conditions. It is also remarkable that the previous derivations also hold for developing flows, such as in the case of the thermosyphon loop studied in Section 4.5. This feature makes the method very versatile and allows us to handle problems which can not be addressed by a periodicity type decomposition, as the one that we used to handle the problem of fully developed isothermal laminar flow in Chapter 3. We will now show how our homogenized model with effective boundary conditions works in practice.
5.7 Numerical Analysis

To show how our homogenized system of equations works in practice, we will apply our approach to the problem which we studied in Chapter 3, where we addressed the modeling of a thermosyphon loop. The presence of the corrugations along the boundaries of the domain introduces an extra degree of difficulty. Also this problem can not be handled by using periodicity arguments and hence we can not reduce the computational domain to just one period. Therefore, the geometry needs to be considered in all of its extension, thereby incurring in higher computational costs for solving the problem numerically. The homogenization approach offers the alternative of avoiding the problem of the multiple scales. In this section we discuss the computational benefits of this approach.

5.7.1 Mesh Generation

Since the domain $\Omega$ of the system (5.8) can not be reduced to just one period, we need to generate a mesh for the domain on all of its extension. The corrugations of the wall introduce multiple length scales and consequently solving the system of equations becomes more difficult. Even the generation of an initial mesh can become computationally demanding for this kind of geometry. In order to show this, we consider a set of examples featuring sinusoidal pipes like the one which was introduced in Section 3.3.2. All the following examples are given in dimensional variables.

For clarity, we recall the expression describing the radius of the sinusoidal pipe, namely

$$R_w(x) = \frac{D}{2} + \frac{a}{2} \left(1 + \sin \left(\frac{2\pi x}{L} - \frac{\pi}{2}\right)\right).$$

(5.82)

where $D$, $a$ and $L$ stand for the diameter, amplitude and period of the pipe respectively. In the following example we consider a fixed diameter $D = 0.02[\text{m}]$ and a fixed ratio between the period and the amplitude of $L/a = 2.5$.

Recalling our description of the radius of the pipe (see Figure 5.2), we see that for this particular example, the small parameter $\epsilon$ and the inner period $L^{\text{in}}$ are

$$\epsilon = \frac{a}{D}, \quad L^{\text{in}} = 2.5.$$

We will vary the amplitude of the pipe and study what are the consequences for mesh generation. We consider $a = \{4 \times 10^{-5}, 2 \times 10^{-5}, 1 \times 10^{-5}\} [\text{m}]$, implying the following values for the small parameter $\epsilon = \{2 \times 10^{-3}, 1 \times 10^{-3}, 5 \times 10^{-4}\}$. 
Figure 5.3: Meshes for a sinusoidal pipe with various values of the small parameter \( \epsilon \) and \( L^\text{in} = 2.5 \). Figure 5.3(a) shows the homogenized domain. All plots are on the same scale and the left boundary is the center line of the geometry. Table 5.1 shows the total amount of mesh elements as well as the time spent on generating them.
In Figure 5.3, we show a region of the domains and their respective meshes for different values of the small parameter $\epsilon$. Figure 5.3(a) shows the homogenized domain. Since the geometry is a simple rectangle, it is straightforward to generate a mesh for this geometry, the mesh can be made finer close to the wall in order to capture boundary layers. On the other hand, generating a mesh for the corrugated geometries is not as simple. Since in the center of the pipe no strong variations are expected, one can try to set a coarse mesh near the center line. However, the small corrugations on the wall require to reduce the mesh size near them so that the edges of the element are able to describe the small details of the geometry. This can be observed from Figures 5.3(b), 5.3(c) and 5.3(d), where the mesh density close the wall increases as $\epsilon$ decreases.

These kind of geometries with multiple scales are not easy to handle by automatic mesh generators. In order to give a quantitative impression of this, in Table 5.1 we show the time that it takes to create an initial mesh when using the default triangular mesh generator "meshinit", from the code COMSOL Multiphysics [22]. The small parameter $\epsilon$ is varied in order to show the importance of the multiple scales. We consider two domains, one with length $L_p = 0.1[\text{m}]$ and another with length $L_p = 0.5[\text{m}]$. The tests presented here, were performed on a computer with an Intel Quad Core 2.4[GHz] processor and 4[GB] of RAM.

Table 5.1 shows the results for two domains, one with length $L_p = 0.1[\text{m}]$ and another with length $L_p = 0.5[\text{m}]$. The time spent on generating the geometries does not increase significantly as we reduce the value of $\epsilon$. In the case of the geometry with $L_p = 0.1[\text{m}]$ for example, the time spent in geometry generation changes from 0.156[s] for $\epsilon = 2 \times 10^{-3}$, to 0.289[s] for $\epsilon = 5 \times 10^{-4}$. However, the required number of mesh elements is more sensitive to changes in $\epsilon$. Roughly speaking, the numbers of mesh elements doubles when the parameter $\epsilon$ is halved. This happens because in order to accurately describe each corrugation, a minimum fixed amount of edges are needed. When the parameter $\epsilon$ is halved, the number of corrugations doubles and consequently also the number of boundary edges at $\Gamma_\epsilon$ doubles. In turn this causes the total amount of elements to increase, for example for $\epsilon = 1 \times 10^{-3}$ and $\epsilon = 5 \times 10^{-4}$, the meshes contain 10466 and 21799 elements, respectively. Therefore the time required for mesh generation is expected to increase significantly. For example from $\epsilon = 2 \times 10^{-3}$ to $\epsilon = 1 \times 10^{-3}$, the time spent on mesh generation increases by a factor of 1.44, while from $\epsilon = 5 \times 10^{-4}$ to $\epsilon = 2.5 \times 10^{-4}$ it increases by a factor of 3.36.

In contrast, the homogenization approach which we introduced above, avoids these complications by solving the equations in the much simpler homogenized domain $\Omega_0$. For example, to generate a quadrilateral mesh for a geometry with 51200 elements, takes only 0.6[s], even though it contains far more elements than any of the other meshes. The time spent to generate the geometry only requires 0.01[s] due to the simple structure of the domain.
Homogenization of the Boussinesq Equations

\[ L_p = 0.1 [m] \]

<table>
<thead>
<tr>
<th>( \epsilon )</th>
<th>( 2 \times 10^{-3} )</th>
<th>( 1 \times 10^{-3} )</th>
<th>( 5 \times 10^{-4} )</th>
<th>( 2.5 \times 10^{-4} )</th>
<th>Homogenized</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometry</td>
<td>0.156[s]</td>
<td>0.21[s]</td>
<td>0.28[s]</td>
<td>0.51[s]</td>
<td>0.01[s]</td>
</tr>
<tr>
<td>Mesh</td>
<td>0.827[s]</td>
<td>2.24[s]</td>
<td>6.96[s]</td>
<td>23.44[s]</td>
<td>0.6[s]</td>
</tr>
<tr>
<td># Elements</td>
<td>4890</td>
<td>10466</td>
<td>21799</td>
<td>43775</td>
<td>51200</td>
</tr>
</tbody>
</table>

\[ L_p = 0.5 [m] \]

<table>
<thead>
<tr>
<th>( \epsilon )</th>
<th>( 2 \times 10^{-3} )</th>
<th>( 1 \times 10^{-3} )</th>
<th>( 5 \times 10^{-4} )</th>
<th>( 2.5 \times 10^{-4} )</th>
<th>Homogenized</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometry</td>
<td>0.39[s]</td>
<td>0.65[s]</td>
<td>–</td>
<td>–</td>
<td>0.01[s]</td>
</tr>
<tr>
<td>Mesh</td>
<td>1.73[s]</td>
<td>65.07[s]</td>
<td>–</td>
<td>–</td>
<td>3.06[s]</td>
</tr>
<tr>
<td># Elements</td>
<td>7563</td>
<td>76395</td>
<td>–</td>
<td>–</td>
<td>256000</td>
</tr>
</tbody>
</table>

Table 5.1: Time spent on geometry and mesh generation for different values of \( \epsilon \) and two domain lengths \( L_p = 0.1 [m] \) and \( L_p = 0.5 [m] \). The geometry considered is a sinusoidal pipe with diameter \( D = 0.02 [m] \), amplitude \( a = \epsilon D \) and period \( L = 2.5a \). The meshes were generated with the default mesh generating function "meshinit" from COMSOL Multiphysics [22]. The entries "–" represent geometries in which "meshinit" is unable to construct a mesh (using the default settings).

Even though so-far we have only considered mesh generation, this simple test already shows one important advantage that mesh generation becomes trivial when solving (5.80) instead of (5.8). We will refer to the method of solving the problem with (5.80), as the homogenization approach. As we can see from Table 5.1, as the overall size of the geometry increases (when \( L_p \) increases), and as the difference in the geometry scales becomes more prominent (when \( \epsilon \) decreases compared to \( R_0^3 \)), the minimum number of mesh elements to discretize the geometry increases and generating a mesh can become very time consuming. Another aspect that one should keep in mind, is that in order to be able to obtain a mesh independent solution for the geometry with multiple scales, further refinement might be required in order to correctly solve for the vortices that might appear inside the corrugations. In turn, this would increase the size of the discrete system of equations and hence the CPU time. The homogenization approach, however, does not suffer from this, and can be used to tackle problems which otherwise would be way too time consuming when solved directly. One could expect this to be the situation when handling an extended geometry in which the multiple scales differ by several order of magnitude.
5.7.2 A Geometry with Small $\epsilon$

Now we discuss the accuracy of the homogenization approach, i.e., we want to see in practice how the solution of the homogenized system compares to the solution of the original problem. With this goal in mind, on the one hand, we solve numerically the system of equations (5.8) in the domain with multiple scales and with the original boundary conditions. On the other hand, we solve the homogenized system of equations (5.80), provided with the effective boundary conditions (5.81).

For the following numerical example, we reconsider the physical properties of methane which we presented in the previous chapter (see Table 4.1). For this particular example, we consider a wall heat transfer coefficient of $\alpha_w = 0.01\, \text{W m}^{-2} \text{K}^{-1}$. The geometry which we will study is a sinusoidal pipe with internal radius $R_w^0 = 0.01\, \text{m}$ and with fixed

We will consider different values for the amplitude $a$, period $L$ and total length $L_p$. We start with the numerical results for a geometry with amplitude $a = 5 \times 10^{-4}\, \text{m}$, period $L = 0.5a$ and length $L_p = 0.1\, \text{m}$. For this geometry, the small parameter is $\epsilon = a/2R_w^0 = 0.025$.

We start by comparing the pressure, temperature and velocity fields obtained by solving the original system of equations (5.8) with the ones obtained by solving the homogenized system of equations (5.80). In addition, we present the solution which is obtained when using the simplistic approach of directly replacing the curved boundary $\Gamma_\epsilon$, by the straight boundary $\Gamma_0$ without constructing equivalent boundary conditions. For convenience, we will refer to these three approaches as the homogenized solution, the full numerical solution and the reduced numerical solution, respectively.

In Figure 5.4 we have plotted the temperature and velocity fields obtained with the three approaches. For convenience, we have plotted the temperature difference $T - T_{\text{ref}}$. The plots displaying the modulus of the velocity $|u|$ obtained from the full, homogenized and reduced numerical solutions are displayed in Figures 5.4(d), 5.4(e) and 5.4(f), respectively. The full numerical solution, which is displayed in Figure 5.4(d), shows a thin high velocity region close to the right corner at the inlet of the pipe which rapidly reaches the center line. The high velocity region quickly shifts towards the center line and as we come close to the outlet it shifts towards the wall. Both, the homogenized solution (Figure 5.4(e)) and the reduced solution (Figure 5.4(f)) show a similar pattern. However, the reduced solution strongly underestimates the modulus of the velocity field. This can be observed at the top right corner of the plots. The homogenized solution does a much better job approximating the full numerical solution, although it still slightly underestimates the modulus of the velocity.

The temperature fields obtained from the full, homogenized and reduced numerical
solutions are displayed in the top Figures 5.4(a), 5.4(b) and 5.4(c), respectively. The full numerical solution in Figure 5.4(a) exhibits the appearance of a thermal boundary layer along the wall of the pipe. The boundary layer is perturbed by the presence of the corrugated wall and this causes small oscillations in the temperature close to the corrugated wall. The corrugations also cause the heat transfer through the wall to increase significantly, compared to a straight wall. We can confirm this by comparing the full solution with the reduced solution in Figure 5.4(c). The full numerical solution shows a region of high temperature displayed in red color on the top right corner, while the reduced solution barely reaches the orange color. In contrast, the homogenized solution is able to capture this high temperature region very well. This shows the importance of the effective boundary condition for the temperature (5.53).

In order to get a more detailed impression of how well the homogenized solution approximates the solution of the original problem, we show how both solutions compare along a vertical line close to the wall and along a line far from the wall. More specifically, Figures 5.5 and 5.6, show the plots of the temperature $T$, pressure $p$ and velocity fields, along the lines $r = 0.0098[m]$ and $r = 0.0085[m]$, respectively. In these figures, the homogenized solutions are displayed in solid red lines. The full numerical solutions are displayed in solid light blue lines. The dash-dot green lines, show the reduced numerical solutions, i.e., the ones obtained by replacing the curved boundary $\Gamma_\epsilon$, by the straight boundary $\Gamma_0$.

In Figure 5.5(a), we can observe how the temperature $T$ obtained from the full numerical solution shows some small fluctuations which occur due to the presence of the corrugated wall. The homogenized solution correctly shows the slow scale behavior of the temperature. Of course the oscillations are not present in the homogenized solution, as these are smoothed out by the homogenization process, however, the accuracy of the method is remarkable. On the other hand, the reduced numerical solution, clearly underestimates the temperature and it does not correctly describe the full numerical solution. The reduced approach is only accurate close to the inlet $x = 0[m]$, where a Dirichlet boundary condition for the temperature is imposed. However, as we go along the axial direction, we can see that this approach gives a wrong answer. From a physical point of view this is expected as just replacing the curved boundary by a straight wall does not take into account the effects that the presence of a curved boundary has on the heat flux. Hence the importance of the effective boundary condition for the temperature (5.53).

The pressure $p$ is displayed in Figure 5.5(b). The full numerical solution shows fast oscillations which are more visible than in the plot of the temperature. Despite these oscillations, the homogenized solution is able to capture the slow scale behavior of the pressure over the whole domain and the homogenized solution passes through the middle of the oscillating solution. In contrast, the pressure obtained with the reduced numerical solution yields an approximation that is only valid close to the ends of the pipe,
Figure 5.4: Temperature $T - T_{ref}$ and velocity modulus $|u|$ for a geometry with amplitude $a = 5 \times 10^{-4}$[m], period $L = 0.5a$ and heat transfer coefficient $\alpha_w = 0.01$[W m$^{-2}$ K]. Figures 5.4(a), 5.4(b) and 5.4(c), show the temperature fields obtained from the full, homogenized and reduced solutions, respectively. The modulus of the velocity for the reduced solutions, are displayed in Figures 5.4(d), 5.4(e) and 5.4(f), and respectively.
Homogenization of the Boussinesq Equations

Figure 5.5: Plot of the temperature $T$, pressure $p$ and axial $u_x$ and radial $u_r$ velocity fields along the line $r = 0.0098$[m]. The red lines show the homogenized solutions. The light blue lines show the full numerical solutions for a geometry with amplitude $a = 5 \times 10^{-4}$[m] (i.e., $\epsilon = 0.025$) and period $L = 0.5a$. For reference, the dash-dot green lines show the reduced numerical solutions, i.e., the problem in which the original curved boundary is directly replaced by a flat boundary without using the effective boundary conditions (5.81).

where boundary conditions of Dirichlet type are imposed. The oscillations of both temperature and pressure, are not captured by the homogenized solution. This is natural as this is one of the main goals of the homogenization procedure. For example when dealing with problems with rapidly oscillating coefficients, the main idea behind homogenization, is to replace these coefficients by an effective coefficient which correctly describes the solution [19]. The importance of the effective boundary conditions (5.81) becomes clear when we look at the green dash-dot line, which shows the solution that is obtained by replacing the curved boundary $\Gamma_\epsilon$ by the flat boundary $\Gamma_0$. As we see from this figure, this simplistic approach is not able to describe the slow scale behavior of the pressure nor the temperature.

In Figure 5.5(c), we display the axial velocity $u_x$. In this case, both approximations nicely follow the solution to the original problem. Finally, we consider the radial velocity field $u_r$ displayed in Figure 5.5(d). In this case the difference between the homogenized and
the original solution becomes more significant than for the other variables of interest. Still the homogenized solution does a better job than the reduced solution for approximating the full numerical solution. The reason for the discrepancies seems to originate from a high sensitivity of the radial velocity to the presence of corrugations in the wall.

In general, we can say that both approaches do a good job for approximating the velocity field. However, the temperature and pressure fields are only well approximated by the homogenized solution, while the reduced numerical solution clearly fails.

Now we proceed to analyze the solutions slightly away from the wall. In Figure 5.6 we show the solutions to the original problem (light blue line), to the homogenized problem (red line) and the reduced solution (dash-dot green line). In Figure 5.6(a) we can see how the oscillations of the full numerical solution are much smaller that the oscillations which appeared close to the wall. The homogenized solution does a good job approximating this behavior while the reduced numerical solution shows significant discrepancies as one approaches the outlet of the pipe. In a similar way as for the temperature, the oscillations of the pressure field are no longer present along the line $r = 0.0085$. From Figure 5.6(b), we can see how the pressure obtained from the homogenized solution does an excellent job approximating the pressure. In fact the full numerical solution and the homogenized solution are only slightly distinguishable. In contrast, the reduced numerical solution, fails. Figures 5.6(c) and 5.6(d) show the axial and radial velocity components, respectively. In the case of the axial velocity, all the solutions are very close to each other and are almost indistinguishable. The case of the radial velocity is different and we can notice some discrepancy between the homogenized and the original solution. Nonetheless, the homogenized solution still does a much better job than the reduced solution.

In order to address the accuracy of the method from a global point of view, we now show some averaged quantities which are of interest in engineering applications. Table 5.2 shows the results which are obtained from the full numerical solution, the homogenized solution and the reduced numerical solution. The quantities which are displayed in the table are the computed volume flow rate $Q [\text{m}^3\text{s}^{-1}]$, Reynolds number $Re$, average axial velocity $U_x [\text{ms}^{-1}]$, total heat transferred to the fluid $H [\text{W}]$, average heat flux $q_w [\text{Wm}^{-2}]$, modified Grashoff number $Gr_{mod}$, as well as the maximum temperature $T_{max} [\text{K}]$ and maximum velocity $u_{max} [\text{ms}^{-1}]$ attained on the whole domain. The maximum temperature attained inside the domain $T_{max}$ is an interesting variable as it should be kept below the saturation temperature in order to avoid undesirable boil-off gas inside the thermosyphon $[99]$. The expressions for computing these quantities are given in (4.19), (4.20) and (4.21).

As we can see from Table 5.2, the Reynolds number estimated by the homogenized solution is a very good approximation of the value obtained from the full numerical solution. In terms of the Reynolds number $Re$, the approximation yields a relative error of 3 %,
Figure 5.6: Plot of the temperature $T$, pressure $p$ and axial $u_x$ and radial $u_r$ velocity fields along the line $r = 0.0085$[m]. The red lines show the solutions obtained from solving the homogenized system of equations (5.80). The light blue lines show the solution obtained from solving the original system of equations with parameters $a = 5 \times 10^{-4}$[m] (which corresponds to $\epsilon = 0.025$), $L = 0.5a$ and $L_p = 0.1$[m]. For reference, the dash-dot green lines show the solutions which are obtained when solving a system of equation in which the original curved boundary is directly replaced by a flat boundary without using the effective boundary conditions (5.81).

while the approximation provided by the reduced method yields a relative error of 13%. When we turn our attention to the maximum attained temperature over the whole domain, we see a value of $T_{\text{max}} = 106.9924$[K] for the original solution. The homogenized solution provides an excellent approximation of this value yielding $T_{\text{max}} = 106.9870$[K]. The simple approach provided by the reduced numerical solution also provides a very good approximation with the value $T_{\text{max}} = 106.9798$. However, when we turn our attention to the total amount of heat transferred to the fluid $H$, the differences in accuracy between the methods becomes evident. The full numerical solution gives a total wall heat transfer of $H = 0.0157$[W] while the reduced numerical solution gives a value of $H = 0.0116$[W], i.e., an error of 26%. In contrast, the homogenized solution gives an approximation with a relative error of only 2%, showing with clarity that it is a superior approach.
Table 5.2: Computed values for several variables of interest based on the full numerical solution, the homogenized solution and the reduced numerical solution, which is obtained by directly replacing the wavy wall by a straight wall. The geometry considered in this example has parameters $a = 5 \times 10^{-4}$ and $L = 0.5a$. The wall heat transfer coefficient is $\alpha_w = 0.01 \text{[W m}^{-2} \text{K}^{-1}]$.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Full Solution</th>
<th>Homogenized</th>
<th>Reduced</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Q$</td>
<td>$1.1310 \times 10^{-6}$</td>
<td>$1.1252 \times 10^{-6}$</td>
<td>$1.0001 \times 10^{-6}$</td>
<td>$\text{m}^3 \text{s}^{-1}$</td>
</tr>
<tr>
<td>$Re$</td>
<td>239.9994</td>
<td>232.6088</td>
<td>206.7468</td>
<td>1</td>
</tr>
<tr>
<td>$U_x$</td>
<td>0.0037</td>
<td>0.0036</td>
<td>0.0032</td>
<td>$\text{m s}^{-1}$</td>
</tr>
<tr>
<td>$H$</td>
<td>0.0157</td>
<td>0.0154</td>
<td>0.0116</td>
<td>$\text{[W]}$</td>
</tr>
<tr>
<td>$q_w$</td>
<td>1.8412</td>
<td>1.8065</td>
<td>1.3645</td>
<td>$\text{[W m}^{-2}]$</td>
</tr>
<tr>
<td>$Gr_{mod}$</td>
<td>$5.8850 \times 10^5$</td>
<td>$5.7742 \times 10^5$</td>
<td>$4.3613 \times 10^5$</td>
<td>1</td>
</tr>
<tr>
<td>$T_{max}$</td>
<td>106.9924</td>
<td>106.987</td>
<td>106.9798</td>
<td>$\text{[K]}$</td>
</tr>
<tr>
<td>$</td>
<td>u</td>
<td>_{max}$</td>
<td>0.0048</td>
<td>0.0047</td>
</tr>
</tbody>
</table>

Overall, the previous example shows that the homogenization procedure is very accurate for describing the solution locally, as well as in terms of global averaged variables. The questions that remain to be answered are how sensitive is the method to the value of $\epsilon$ and what are the benefits of using homogenization in terms of computational time.

### 5.7.3 A Geometry with Moderate $\epsilon$

In the previous section we showed how the homogenization approach performs for a geometry with $a = 5 \times 10^{-4} [\text{m}]$ ($\epsilon = 0.025$) and with internal period $L^m = 0.5$. Now we consider a geometry with larger amplitude, namely $a = 5 \times 10^{-3} [\text{m}]$. The period of the pipe has the same form, i.e., $L = 0.5a$. In this case, the ”small” parameter becomes $\epsilon = a/2R_w^o = 0.25$, i.e., 10 times larger than in the example of the previous section. The value for the inner period $L^m = 0.5$ and the value of $\epsilon$, define a geometry which corresponds to the cavity type configuration introduced in Section 3.5.1. As we pointed out before, these geometries constitute a challenge due to the complicated flow structures which develop inside the cavity and it is interesting to test how the homogenized model works for such geometries.

We start by analyzing how well the homogenized solution approximates the full numerical solution of the original problem near the wall. In Figure 5.7, we plot the temperature $T$, pressure $p$, axial velocity $u_x$ and radial velocity $u_r$, along the vertical line $r = 0.09 [\text{m}]$, i.e., 0.01 [m] away from the wall of the homogenized boundary. The solutions obtained from solving the homogenized system (5.80) are plotted in red dashed lines. The solutions obtained from the original curved domain with parameter $a = 5 \times 10^{-3} [\text{m}]$, are displayed with green dashed-dotted lines. The light blue solid lines show the solutions which are obtained from the reduced numerical solution. As we can observe from
Figure 5.7(b), the pressure obtained from our homogenized solution describes very accurately the slow scale behavior of the pressure obtained from the original domain. The green dashed-dotted curve shows rapid oscillations which are stronger than in the previous example. Even though the small parameter is rather large, i.e., $\epsilon = 0.25$, the homogenized solution is still able to describe the slow scale behavior of the pressure and the reduced numerical solution proves to be completely wrong as soon as one departs from the ends of the pipe.

In Figure 5.7(a), we can observe the plot of the temperature $T$. In this case we can observe how the temperature obtained from the homogenized solution is slightly above the temperature obtained from the full numerical solution. The difference between both solutions reduces as one approaches the end of the pipe $L_p = 0.1 \text{[m]}$. The solution obtained with the reduced numerical solution, is only accurate close to the inlet $x = 0 \text{[m]}$, where a Dirichlet boundary condition for the temperature is imposed. However, as we move further in the axial direction, we can see that this approach is totally wrong. From a physical point of view this can be explained because just replacing the curved boundary by a straight one does not take into account the influence of the corrugated wall on the heat flux. The effects of the corrugations are more significant in this case because the large value in the amplitude $a$ induces a larger correction via the effective boundary condition for the heat flux (5.53).

For the axial velocity $u_x$, displayed in Figure 5.7(c), one could say that both the homogenized solution and the reduced numerical solution work reasonably well. Although it is possible to observe certain differences between these approximations near the inlet of the pipe, where we can observe how the blue solid line goes slightly above the homogenized and the real solution. Finally we consider the radial velocity field $u_r$ in Figure 5.7(d). In this case the difference between the homogenized and the original solution becomes more significant than for the other variables of interest. Altogether, we confirm that the homogenized solution does a better job than the reduced numerical solution.

Now we conclude discussing a region far from the wall. In particular, we study the solutions along the line $r = 0.05 \text{[m]}$. In Figure 5.8, we have plotted the pressure, temperature and velocity components obtained from the full numerical solution (green dash-dot line), the homogenized solution (red dash line) and the reduced numerical solution (blue solid line).

In any case, we can see that the homogenized solution works remarkably well, specially considering the fact that $\epsilon = 0.25$, which is quite a large value. For smaller values of $\epsilon$ the homogenized solution is more accurate, as we saw in the previous example. In order to give a complete comparison, in Table 5.3, we present the computed values for the flow rate $Q$, Reynolds number $Re$, total heat transfer $H$ among other variables of interest. The expressions for computing these quantities are given in (4.19), (4.20) and (4.21).
5.7 Numerical Analysis

Figure 5.7: Plot of the temperature $T$, pressure $p$ and axial $u_x$ and radial $u_r$ velocity fields along the line $r = 0.09[m]$. The red dashed lines show the solutions obtained from solving the homogenized system of equations (5.80). The green dash-dotted lines show the full numerical solution obtained from solving the original system of equations with parameter $a = 5 \times 10^{-3}[m]$, $(\varepsilon = 0.25) L = 0.5a$ and $L_p = 0.1[m]$. For reference, the light blue solid line shows the reduced numerical solution, which is obtained when solving a system of equation in which the original curved boundary is directly replaced by a flat boundary without using the effective boundary conditions (5.81).

Compared to the previous example, the estimations of the global quantities are not as accurate. This comes as a result of the large value of $\varepsilon = 0.25$ and the small value of $L = 0.5\varepsilon$. This causes the flow inside the corrugations to be of cavity type, which promotes the appearance of vortices as we discussed in Section 3.5.1. The Reynolds number obtained from the original geometry is $Re = 429.925$ while the value provided by the homogenized solution is $Re = 374.8205$. Therefore, our homogenization approach yields a relative error of 12%, while the approximation provided by the reduced numerical solution yields a relative error of 52%, with the estimated value $Re = 206.7468$. When we turn our attention to the total heat transfer, we find out that the homogenization alternative gives us the estimated value $H = 0.0488$, while from the full numerical solution we obtain $H = 0.0608$, i.e., a relative error of 19% which is not as good as in the previous example, but it is still much better that the value $H = 0.0116$ with a relative
Figure 5.8: Plot of the temperature $T$, pressure $p$ and axial $u_x$ and radial $u_r$ velocity fields along the line $r = 0.05\,[\text{m}]$. The red dashed lines show the solutions obtained from solving the homogenized system of equations (5.80). The green dash-dotted lines show the full numerical solution obtained from solving the original system of equations with parameter $a = 5 \times 10^{-3}\,[\text{m}]$, ($\epsilon = 0.25$), $L = 0.5a$ and $L_{F} = 0.1\,[\text{m}]$. For reference, the light blue solid line shows the reduced numerical solution, which is obtained when solving a system of equation in which the original curved boundary is directly replaced by a flat boundary without using the effective boundary conditions (5.81).

The previous examples confirm that solutions obtained with the homogenization approach can be used to calculate quantities with engineering purposes with a good level of accuracy. In the next section we will proceed to study the accuracy of the method in flow conditions with large wall heat transfer coefficients. Finally we will turn our attention to the computational costs of the homogenization approach and will evaluate the computational advantages of our approach.
Table 5.3: Computed values for several variables of interest based on the full numerical solution, the homogenized solution and the reduced numerical solution, which is obtained by directly replacing the wavy wall by a straight wall. The geometry considered in this example has parameters $a = 5 \times 10^{-3}$ and $L = 0.5a$.

### 5.7.4 Accuracy of the Method for Large Wall Heat Transfer Coefficients

Now we discuss how our homogenization method behaves for higher wall heat transfer coefficients. The present numerical experiments were performed on a geometry with parameters $L_p = 0.1\text{[m]}$, $a = 5e-4\text{[m]}$ and $L^m = 0.5a$. The wall heat transfer coefficient $\alpha_w$ was varied from $0.01\text{[W m}^{-2}\text{K}^{-1}]$ up to $1\text{[W m}^{-2}\text{K}^{-1}]$. In Figure 5.9 we have plotted the Reynolds number, the total heat transferred to the fluid and the maximum attained temperature inside the thermosyphon for the given geometry. For reference, we have included the numerical value of these and other global quantities in Table 5.6, which shows the results obtained with the full numerical solution and with the homogenized solution. The expressions for computing these quantities are given in (4.19), (4.20) and (4.21).

First in Figure 5.9(a) we can observe how the total heat transferred to the fluid increases in a linear fashion as function of the wall heat transfer coefficient. As we can observe from this figure, the homogenized solution does a good job for predicting the total heat flux over the whole range of wall heat transfer coefficients. From the values presented in Table 5.6, we can see that the maximum relative error for the total heat transfer computed with the homogenized solution is less than $2\%$, over the whole range of $\alpha_w$.

The increment in heat transfer which we have seen, induces larger temperature differences within the thermosyphon, which in turn give rise to larger Reynolds numbers. This behavior can be observed in Figure 5.9(b), where we can also see that the dependency of the Reynolds number on the wall heat transfer coefficient $\alpha_w$ is non-linear. The homogenized solution still does a good job predicting the Reynolds number. The approximation obtained with the homogenized solution remains accurate over the whole range of values for the wall heat transfer coefficient, see Table 5.6. For example for
Homogenization of the Boussinesq Equations

**Full Solution**

<table>
<thead>
<tr>
<th>( \alpha_w )</th>
<th>0.01</th>
<th>0.05</th>
<th>0.1</th>
<th>0.5</th>
<th>1</th>
<th>([\text{W m}^{-2}\text{K}^{-1}])</th>
</tr>
</thead>
<tbody>
<tr>
<td>( Q )</td>
<td>(1.1576 \times 10^{-6} )</td>
<td>(2.2638 \times 10^{-6} )</td>
<td>(3.0087 \times 10^{-6} )</td>
<td>(5.7886 \times 10^{-6} )</td>
<td>(7.6562 \times 10^{-6} )</td>
<td>([\text{m}^3\text{s}^{-1}])</td>
</tr>
<tr>
<td>( \text{Re} )</td>
<td>239.3002</td>
<td>467.9651</td>
<td>621.9661</td>
<td>(1.1966 \times 10^3 )</td>
<td>(1.5827 \times 10^3 )</td>
<td>([1] )</td>
</tr>
<tr>
<td>( U_x )</td>
<td>0.0037</td>
<td>0.0072</td>
<td>0.0096</td>
<td>0.0184</td>
<td>0.0244</td>
<td>([\text{m} \text{s}^{-1}])</td>
</tr>
<tr>
<td>( H )</td>
<td>0.0157</td>
<td>0.0780</td>
<td>0.1551</td>
<td>0.7680</td>
<td>1.5373</td>
<td>([\text{W}])</td>
</tr>
<tr>
<td>( q_w )</td>
<td>1.8447</td>
<td>9.1463</td>
<td>18.1967</td>
<td>90.0859</td>
<td>180.3196</td>
<td>([\text{W m}^{-2}])</td>
</tr>
<tr>
<td>( \text{Gr}_{\text{mod}} )</td>
<td>(5.8962 \times 10^5 )</td>
<td>(2.9234 \times 10^6 )</td>
<td>(5.8162 \times 10^6 )</td>
<td>(2.8794 \times 10^7 )</td>
<td>(5.7636 \times 10^7 )</td>
<td>([\text{]})</td>
</tr>
<tr>
<td>( T_{\text{max}} )</td>
<td>106.9925</td>
<td>107.1046</td>
<td>107.2222</td>
<td>107.9819</td>
<td>108.7989</td>
<td>([\text{K}])</td>
</tr>
<tr>
<td>( T_{\text{max}}^* )</td>
<td>106.9869</td>
<td>107.0789</td>
<td>107.1717</td>
<td>107.7336</td>
<td>108.2977</td>
<td>([\text{K}])</td>
</tr>
<tr>
<td>(</td>
<td>u</td>
<td>_{\text{max}} )</td>
<td>0.0048</td>
<td>0.0087</td>
<td>0.0114</td>
<td>0.0232</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>( \alpha_w )</th>
<th>0.01</th>
<th>0.05</th>
<th>0.1</th>
<th>0.5</th>
<th>1</th>
<th>([\text{W m}^{-2}\text{K}^{-1}])</th>
</tr>
</thead>
<tbody>
<tr>
<td>( Q )</td>
<td>(1.1253 \times 10^{-6} )</td>
<td>(2.191 \times 10^{-6} )</td>
<td>(2.0994 \times 10^{-6} )</td>
<td>(5.5913 \times 10^{-6} )</td>
<td>(7.4074 \times 10^{-6} )</td>
<td>([\text{m}^3\text{s}^{-1}])</td>
</tr>
<tr>
<td>( \text{Re} )</td>
<td>232.6118</td>
<td>453.1054</td>
<td>601.4249</td>
<td>(1.1558 \times 10^3 )</td>
<td>(1.5312 \times 10^3 )</td>
<td>([1] )</td>
</tr>
<tr>
<td>( U_x )</td>
<td>0.0036</td>
<td>0.0070</td>
<td>0.0093</td>
<td>0.0178</td>
<td>0.0236</td>
<td>([\text{m} \text{s}^{-1}])</td>
</tr>
<tr>
<td>( H )</td>
<td>0.0154</td>
<td>0.0769</td>
<td>0.1538</td>
<td>0.7696</td>
<td>1.5383</td>
<td>([\text{W}])</td>
</tr>
<tr>
<td>( q_w )</td>
<td>1.8066</td>
<td>9.0422</td>
<td>18.0935</td>
<td>90.5203</td>
<td>180.9308</td>
<td>([\text{W m}^{-2}])</td>
</tr>
<tr>
<td>( \text{Gr}_{\text{mod}} )</td>
<td>(5.77 \times 10^5 )</td>
<td>(2.890 \times 10^6 )</td>
<td>(5.783 \times 10^6 )</td>
<td>(2.8933 \times 10^7 )</td>
<td>(5.7831 \times 10^7 )</td>
<td>([\text{]})</td>
</tr>
<tr>
<td>( T_{\text{max}} )</td>
<td>106.9870</td>
<td>107.0793</td>
<td>107.1724</td>
<td>107.7370</td>
<td>108.3075</td>
<td>([\text{K}])</td>
</tr>
<tr>
<td>(</td>
<td>u</td>
<td>_{\text{max}} )</td>
<td>0.0047</td>
<td>0.0085</td>
<td>0.0113</td>
<td>0.0227</td>
</tr>
</tbody>
</table>

**Table 5.4:** Computed values of several variables of interest for various values of the wall heat transfer coefficient \( \alpha_w \). The top table shows the values based on the full numerical solution and the bottom table the ones based on the homogenized solution. The geometry considered is a sinusoidal pipe with amplitude \( a = 5 \times 10^{-4} \), period \( L = 0.5 \epsilon \) (i.e., \( L^{bn} = 0.5 \)), length \( L_p = 0.1 \text{[m]} \), diameter \( D = 2 \text{[cm]} \).
5.7 Numerical Analysis

Figure 5.9: Plot of the total heat transferred $H$ and resulting Reynolds number $Re$. The blue lines show the full numerical solutions, while the red dashed lines show the homogenized solutions. The geometry has the parameters $a = 5 \times 10^{-4}$[m], $(\varepsilon = 0.025)$, $L = 0.5a$ and $L_p = 0.1$[m].

For $\alpha_w = 0.01$[W m$^{-2}$ K$^{-1}$], the Reynolds number obtained from the full and homogenized solutions are $Re = 239.3002$ and $Re = 232.6118$, respectively. This translates into a relative error of 2.7%. The relative error increases slightly when $\alpha_w = 1$[W m$^{-2}$ K$^{-1}$], where the accuracy of the homogenized method is a relative error of 3.25%. Over the whole range of values for $\alpha_w$ presented here, the maximum relative error of the Reynolds number computed with the homogenized solution is only 3.4%.

The previous discussion shows that the homogenization approach is able to accurately predict averaged type quantities such as the Reynolds number $Re$ and the total heat transfer $H$. Now we turn our attention to the maximum attained temperature $T_{max}$. The maximum temperature computed from the full numerical solution is displayed with the solid blue line in Figure 5.9(c). As we observe from this figure, the maximum temperature computed from the homogenized solution (red dash line) deviates from the one obtained from the full numerical solution. At first this might look as some inaccuracy of the method, however this is not the case. The observed difference in temperature, comes as a result of the fact that the homogenized domain does not contain the corrug-
Homogenization of the Boussinesq Equations

ations and hence, the maximum temperature computed with the homogenized solution corresponds to the maximum temperature in the homogenized domain $\Omega_0$ only. Keeping this in mind, from the full numerical solution, we computed the restricted maximum temperature, defined as

$$T_{\text{max}}^* := \max_{r < 0.01\text{[m]}} T.$$

This restricted maximum temperature is displayed in Figure 5.9(c) in green with triangles pointing down. The homogenized solution predicts very accurately the restricted maximum temperature, we can confirm this in Table 5.6. It is remarkable that the maximum error for the restricted maximum temperature is below 0.1% over the whole range of $\alpha_w$ studied here.

In conclusion, we can say that the homogenization method remains accurate also for flow conditions with large Reynolds numbers. In our example we consider values up to $Re = 1582$. The method is useful for predicting global quantities with engineering purposes such as the total heat transfer $H$ and the volume flow rate $Q$, but is also useful for predicting quantities such as the maximum temperature.

5.7.5 Speed Up of the Homogenization Approach

Finally we consider the computational advantages of the homogenization method. In Section 5.7.1 we already discussed how mesh generation becomes straightforward with the homogenization approach, while directly discretizing the geometry suffers from the fact that the number of mesh elements doubles when the size of the corrugations is halved. This simple fact immediately reflects on the time spend for solving the discrete system of equation.

In Table 5.5 we can observe the CPU time for computing the full numerical solutions and the homogenized solutions for geometries with length $L_p = 0.1\text{[m]}$ and various values for the parameters $a$ and $L_{in}$. The first thing that we can observe is the significant increment in computation time as the size of the corrugation decreases. For $L_{in} = 3$, the time increases by a factor of 5 from $142\text{[s]}$ for $a = 8.33 \times 10^{-4}\text{[m]}$ to $705\text{[s]}$ for $a = 4.166 \times 10^{-4}$. This effect becomes more prominent for corrugations of cavity type, for example, for $L_{in} = 0.5$ the time required to compute the full numerical solution increases by a factor 20 when $a$ is halved. The homogenized solution can be computed in just a fraction of the time required to obtain the full numerical solution. For example, for $L_{in} = 1$ and $a_{in} = 2.5 \times 10^{-3}$. the homogenization approach is 5 times faster than the full numerical approach. The speed up becomes much more prominent for smaller corrugation sizes, for example for $L_{in} = 1$ and $a = 1.25 \times 10^{-3}$, the homogenization approach is 87 times faster than the direct approach, showing a great reduction in computation time.

We also discuss the effect of the wall heat transfer coefficient $\alpha_w$ on the computational
### Table 5.5: CPU time for computing the full numerical solution and homogenized solution for a sinusoidal pipe of length $L_p = 0.1\text{[m]}$, wall heat transfer coefficient $\alpha_w = 0.01\text{[W m}^{-2}\text{K}^{-1}]$, diameter $D = 2\text{[cm]}$, aspect ratio $L_e = 0.5$ and several values for the amplitude $a$ and for the parameter $L_{in}$, where the period of the corrugations can be written as $L = L_{in}a$.

<table>
<thead>
<tr>
<th>$L_{in}$</th>
<th>$a$[m]</th>
<th>Time [s]</th>
<th>Accumulated [s]</th>
<th>Homogenized</th>
<th>Accumulated [s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>$5 \times 10^{-3}$</td>
<td>303.0486</td>
<td>6199.7</td>
<td>73.07</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$2.5 \times 10^{-3}$</td>
<td>6199.7</td>
<td>73.07</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>$2.5 \times 10^{-3}$</td>
<td>299.8644</td>
<td>5395</td>
<td>62.4611</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$1.25 \times 10^{-3}$</td>
<td>5395</td>
<td>62.4611</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.5</td>
<td>$1 \times 10^{-3}$</td>
<td>195.1992</td>
<td>799</td>
<td>59.16</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$5 \times 10^{-4}$</td>
<td>799</td>
<td>59.16</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>$8.333 \times 10^{-4}$</td>
<td>142.5057</td>
<td>705.5219</td>
<td>56.89</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$4.1666 \times 10^{-4}$</td>
<td>705.5219</td>
<td>56.89</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>$6.25 \times 10^{-4}$</td>
<td>105.3873</td>
<td>588.9635</td>
<td>53.75</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$3.125 \times 10^{-4}$</td>
<td>588.9635</td>
<td>53.75</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Table 5.6: Variation of the computation time as function of the wall heat transfer coefficient $\alpha_w$. The other parameters are $a = 5 \times 10^{-4}$, $L_p = 0.1\text{[m]}$, $L_{in} = 2.5$, $D = 2\text{[cm]}$ and $L_e = 0.5$.

<table>
<thead>
<tr>
<th>$\alpha_w$[W m}^{-2}\text{K}^{-1}]</th>
<th>0.01</th>
<th>0.05</th>
<th>0.1</th>
<th>0.5</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full [s]</td>
<td>1580</td>
<td>1153</td>
<td>755</td>
<td>1661</td>
<td>1265</td>
</tr>
<tr>
<td>Accumulated [s]</td>
<td>1580</td>
<td>2733</td>
<td>3488</td>
<td>5149</td>
<td>6414</td>
</tr>
<tr>
<td>Homogenized [s]</td>
<td>341</td>
<td>132</td>
<td>120</td>
<td>189</td>
<td>163</td>
</tr>
<tr>
<td>Accumulated [s]</td>
<td>341</td>
<td>473</td>
<td>593</td>
<td>782</td>
<td>945</td>
</tr>
</tbody>
</table>
Homogenization of the Boussinesq Equations

\[ L_{in} = 1 \quad a = 3.75 \times 10^{-3} \]

<table>
<thead>
<tr>
<th>( L_p )</th>
<th>Full</th>
<th>Homogenized</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.1</td>
<td>62</td>
</tr>
<tr>
<td>Units</td>
<td>0.3</td>
<td>132</td>
</tr>
<tr>
<td></td>
<td>0.75</td>
<td>158</td>
</tr>
<tr>
<td></td>
<td>42569</td>
<td>[s]</td>
</tr>
<tr>
<td></td>
<td>308</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1361</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>( L_{in} = 2 )</th>
<th>( a = 3.75 \times 10^{-3} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full</td>
<td>Homogenized</td>
</tr>
<tr>
<td>0.1</td>
<td>62</td>
</tr>
<tr>
<td>0.3</td>
<td>132</td>
</tr>
<tr>
<td>0.75</td>
<td>158</td>
</tr>
<tr>
<td>27325</td>
<td>[s]</td>
</tr>
<tr>
<td>211</td>
<td></td>
</tr>
<tr>
<td>695</td>
<td></td>
</tr>
</tbody>
</table>

Table 5.7: Variation of the CPU time as function of the total length of the pipe \( L_p \). We consider various values for the parameter \( L_{in} \) and \( a \). The other parameters are \( \alpha_w = 0.01 \text{[W m}^{-2} \text{K}^{-1}] \), \( D = 2 \text{[cm]} \) and \( L_c = 0.5 \).

This speed up in computational time becomes crucial when dealing with larger domains, i.e., with larger \( L_p \). We address this briefly in Table 5.7, where we show how the computation costs of the full numerical solution increase much faster than for the homogenized solution for increasing \( L_p \). For example, for the geometry with \( L_{in} = 2 \) and \( a = 3.75 \times 10^{-3} \text{[m]} \), the homogenized method is about 4 times faster than the full numerical approach. When the length of the pipe is increased to \( L_p = 0.75 \text{[m]} \), the speed up factor is roughly 175, which shows the benefits of the homogenized model.

As a main conclusion we see that the homogenization method accurately predicts the solution in a fraction of the costs of the full numerical approach. Also after the previous discussion it is not difficult to imagine a practical situation in which applying the full numerical approach becomes extremely time consuming. In this case the homogenization method gives us the opportunity to compute a solution that otherwise would be unfeasible to compute.
Chapter 6

Isothermal Turbulent Flow

In this final chapter we consider the problem of simulating isothermal turbulent flow. We start with a brief introduction to turbulence modeling and present the Reynolds averaged Navier-Stokes (RANS) equations. Next we discuss the problem of the closure for turbulence modeling. We concentrate our attention in the algebraic LVEL model and in the two-equation $k$-$\varepsilon$ model. We focus on the applicability of these models for estimating the friction factor in turbulent flows. For the mathematical derivation of the RANS we refer the reader to [66], for a more comprehensive review on the far from resolved topic of turbulence, we refer the reader to [126].

6.1 Turbulence Modeling

Many practical problems in engineering involve flows that are turbulent, and thus it is important to understand how turbulence affects the flow and in particular the wall shear stress. Turbulent flow is characterized by rapid fluctuations of swirling regions of fluid, called eddies. These fluctuations provide an additional mechanism for both, momentum and energy transfer. Turbulence is a very complex mechanism dominated by fluctuations, and despite the huge amount of work done in this area, turbulence is far from being well understood [15].

One approach which has given rise to several turbulence models is the Reynolds averaged Navier-Stokes equations or RANS. This is a statistical approach which consists of looking at the turbulent fluctuations of the flow variables as if they were random. There are also other approaches to tackle the problem of turbulence, such as Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES). As the name suggest, the DNS ap-
Isothermal Turbulent Flow

The approach is aimed at solving the full time-dependent Navier-Stokes equations (see (2.37)) without the help of any turbulence model. This means that the whole range of spatial and temporal scales of the turbulence must be resolved. This restricts the usage of DNS to moderate Reynolds numbers \[71, 126\]. The LES approach is a mathematical model for turbulence which is aimed to reduce the computational costs associated with DNS. This is achieved by applying low-pass filtering to the Navier-Stokes equations for eliminating small scales of the solution [89]. Despite this fact, the computational costs associated with LES methods are greater than those associated with RANS methods [66, pp.108]. Therefore we restrict ourselves to RANS models.

### 6.1.1 Reynolds Averaged Navier-Stokes Equations (RANS)

Below we present the derivation of the Reynolds averaged Navier-Stokes equations or RANS. The present derivation is based on [66]. The RANS require, as the name suggests, averaging. There are three prominent forms of averaging commonly used in turbulence modeling research, these are time averaging, spatial averaging and the ensemble averaging. Spatial averaging is appropriate for homogeneous turbulence, i.e., a turbulent flow which on average is uniform in all directions. The ensemble averaging is the most general type of averaging and it is constructed as the arithmetic average of a certain number of identical experiments. These experiments can either be physical or numerical. In our case, we will derive the RANS equations by applying time averaging. Time averaging allows us to describe inhomogeneous turbulence in a convenient way.

For stationary turbulence, i.e., a turbulent flow that on average does not vary in time, the time average of the instantaneous pressure \(p(x, t)\) (also called mean pressure), for example, is defined as

\[
\bar{p}(x) = \lim_{t \to \infty} \frac{1}{t} \int_{0}^{t} p(x, \theta) d\theta.
\] (6.1)

In a similar way we can define the averaged velocity, axial velocity, radial velocity, and azimuthal velocity, denoted by \(\bar{u}, \bar{u}_x, \bar{u}_r, \bar{u}_\phi\), respectively. We note that the limits of integration are arbitrary and they can be shifted. In this definition it is also clear that the average can no longer be a function of time. From this fact we conclude that all time derivatives of averaged variables, such as \(\bar{u}_x\) or \(\bar{u}_r\), are identically \(0\).

Now we introduce the Reynolds decomposition [100]. The idea of the averaging procedure for the Navier-Stokes equations relies on decomposing all instantaneous flow variables as the sum of a mean (indicated by an overline \(\bar{\cdot}\)) and a fluctuating part (indicated by a prime \(\cdot'\)). For the pressure, for example, we have

\[
p(x, t) = \bar{p}(x) + p'(x, t),
\] (6.2)
where $\overline{p}$ is defined as in (6.1) and $p'$ denotes the pressure fluctuation. A key observation is that the averaged quantities $\overline{p}$, $\overline{u_x}$ and $\overline{u_r}$ are all independent of time.

### 6.1.2 Properties of Time Averaging

The time averaging operation $\overline{\cdot}$ has certain properties which are useful for deriving the RANS equations. Let $\varphi_1$ and $\varphi_2$ denote two instantaneous flow variables, then it is easy to prove the following properties

\begin{align}
\varphi_1 + \varphi_2 &= \overline{\varphi_1 + \varphi_2}, \quad \overline{\varphi_1} = \varphi_1, \quad (\overline{\varphi_1})' = 0, \quad \overline{\varphi_1} = a \overline{\varphi_1}, \quad \frac{\partial \overline{\varphi_1}}{\partial x_i} = \frac{\partial \varphi_1}{\partial x_i}, \quad (6.3)
\end{align}

where $a = a(x)$ is a time-independent function and $x_i$ denotes an arbitrary spatial coordinate. It is also easy to see that the gradient $\nabla$ and divergence $\nabla \cdot$ operators inherit the last property. When we take the time average of the product of two quantities we have the following

\begin{align}
\overline{\varphi_1 \varphi_2} = (\overline{\varphi_1 + \varphi_1'}) (\overline{\varphi_2 + \varphi_2'}) = \overline{\varphi_1 \varphi_2} + \overline{\varphi_1 \varphi_2'} + \overline{\varphi_1' \varphi_2} + \overline{\varphi_1' \varphi_2'}, \quad (6.4)
\end{align}

where the last equality follows from the properties that the time average of a mean variable is equal to itself and that the time average of a fluctuating variable is 0; see (6.3). However, the time average of the product of two fluctuating quantities is not zero, in fact, as we will show now, these terms are the ones responsible for the appearance of the Reynolds stresses.

### 6.1.3 Derivation of the RANS Equations

Next we derive the RANS equations. First we need to recall the isothermal incompressible Navier-Stokes equations (3.17). These are constituted by the continuity equation (2.37a) and the momentum equations (2.37b). Omitting the heat equation (since we will treat isothermal turbulence) and replacing $\rho_{ref}$ by $\rho$ (for notational convenience), the system in dimensional form reads

\begin{align}
\nabla \cdot \mathbf{u} &= 0, \quad (6.5a)

\partial_t \mathbf{u} + \mathbf{u} \cdot \nabla \mathbf{u} &= -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u}, \quad (6.5b)
\end{align}

where $\mathbf{u}$, $p$ and $\nu$ stand for the velocity, the pressure and the kinematic viscosity, respectively. The constant body force term is not included. We start by substituting the
Reynolds decomposition for the velocity, i.e., \( u = \overline{u} + u' \), into (6.5a). We obtain
\[
\nabla \cdot u = \nabla \cdot (\overline{u} + u') = \nabla \cdot \overline{u} + \nabla \cdot u' = 0, \tag{6.6}
\]
after time averaging this equation we arrive at
\[
\nabla \cdot \overline{u} + \nabla \cdot \overline{u'} = 0, \tag{6.7}
\]
and using (6.3), we obtain the divergence free equation for the mean velocity \( \overline{u} \), namely
\[
\nabla \cdot \overline{u} = 0. \tag{6.8}
\]
Substituting (6.8) in (6.6), we find that the velocity fluctuations \( u' \) are also divergence free,
\[
\nabla \cdot u' = 0. \tag{6.9}
\]
Now we proceed with the momentum equation (6.5b). We again make use of the Reynolds decomposition, for the velocity \( u \) and for the pressure \( p \). In this case we obtain
\[
\partial_t (\overline{u} + u') + (\overline{u} + u') \cdot \nabla (\overline{u} + u') = -\frac{1}{\rho} \nabla (\overline{p} + p') + \nu \nabla^2 (\overline{u} + u'). \tag{6.10}
\]
By definition \( \overline{u} \) is independent of \( t \) and we can eliminate the first term from (6.10). Expanding the dot product on the left hand side we obtain
\[
\partial_t u' + \overline{u} \cdot \nabla u' + u' \cdot \nabla \overline{u} + u' \cdot \nabla u' = -\frac{1}{\rho} \nabla (\overline{p} + p') + \nu \nabla^2 (\overline{u} + u'). \tag{6.11}
\]
Taking the time average of (6.11) and using the fact that time averaging and spatial differentiation commute, we obtain
\[
\overline{\partial_t u'} + \overline{u} \cdot \overline{\nabla u'} + \overline{u'} \cdot \overline{\nabla u} = -\frac{1}{\rho} \nabla (\overline{p} + p') + \nu \nabla^2 (\overline{u} + u'). \tag{6.12}
\]
By using the linearity of the averaging operator and (6.3), one can easily prove that \( \overline{u} \cdot \overline{\nabla u'} \) and \( \overline{u'} \cdot \overline{\nabla u} \) are both identically 0. The time average of the unsteady term \( \partial_t u' \) also becomes 0. To see this we apply the fundamental theorem of calculus to the integral defining \( \overline{\partial_t u'} \) (see (6.1) and obtain
\[
\overline{\partial_t u'} = \lim_{t \to \infty} \frac{u'(x, t) - u'(x, 0)}{t} = 0, \tag{6.13}
\]
where the last equality follows from the assumption that the fluctuations \( u' \) are bounded. Hence we can eliminate the unsteady term in (6.12). In addition, the time averages \( \overline{p'} \)
and \( \overline{u} \) drop out by using (6.3). Thus we arrive at the equation
\[
\overline{u} \cdot \nabla \overline{u} + \overline{u}' \cdot \nabla \overline{u}' = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \overline{u}.
\] (6.14)

In order to rewrite the terms on the left hand side of (6.14), we need to make use of the identity \( \nabla \cdot (ab^T) = b \nabla \cdot a + a \cdot \nabla b \) and use the fact that \( u' \) is divergence free. Using these conditions, we conclude that
\[
\overline{u}' \cdot \nabla \overline{u}' = \nabla \cdot (\overline{u}' \overline{u}'^T).
\] (6.15)

Finally, changing the spatial differentiation with the time averaging, using (6.3) and multiplying the equation by \( \rho \), we obtain the Reynolds averaged Navier-Stokes momentum equations, which read
\[
\rho \overline{u} \cdot \nabla \overline{u} = -\nabla p + \mu \nabla^2 \overline{u} + \nabla \cdot \left( -\rho \overline{u}' \overline{u}'^T \right).
\] (6.16)

The first thing that one can notice is that (6.16) contains more unknowns than equations. The extra unknowns appear in the term \( -\rho \overline{u}' \overline{u}'^T \). Even though this term is not stress at all, its units are those of stress. Despite the fact that this terms is just a re-worked version of the contribution of the fluctuations via the non-linear advection term, as far as the motion is concerned, these terms can be treated as if they were stresses. That is why the terms \( -\rho \overline{u}' \overline{u}'^T \) are referred to as the Reynolds stresses. We will denote the Reynolds stress tensor by \( \mathcal{R} \). The matrix representation of the Reynolds stress tensor in Cartesian coordinates is the following
\[
\mathcal{R} := -\rho \overline{u}' \overline{u}'^T = -\rho \begin{pmatrix}
\overline{u}_1^2 & \overline{u}_1 \overline{u}_2 & \overline{u}_1 \overline{u}_3 \\
\overline{u}_2 \overline{u}_1 & \overline{u}_2^2 & \overline{u}_2 \overline{u}_3 \\
\overline{u}_3 \overline{u}_1 & \overline{u}_3 \overline{u}_2 & \overline{u}_3^2
\end{pmatrix},
\] (6.17)

where \( \overline{u}_i' \) denotes the \( i \)-th Cartesian component of the velocity \( \overline{u}' \). The trace of the Reynolds stress tensor can be written as
\[
\text{tr} (\mathcal{R}) = -2 \rho k,
\] (6.18)

where \( k [\text{J kg}^{-1}] \) is the specific turbulent kinetic energy, which is defined as
\[
k := \frac{1}{2} \overline{|\overline{u}'|^2} = \frac{1}{2} |\overline{u}'|^2.
\] (6.19)
Summarizing, the RANS equations which we have derived are

\[ \nabla \cdot \mathbf{u} = 0, \tag{6.20a} \]
\[ \rho \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u} + \nabla \cdot \mathbf{R}, \tag{6.20b} \]

were the Reynolds stress tensor \( \mathbf{R} \) needs to be modeled. Before proceeding, we briefly discuss time-dependent RANS.

### 6.1.4 Time-Dependent RANS

The most general form of the RANS equations typically includes the transient term \( \partial_t \mathbf{u} \). However, the derivation which we made showed that \( \mathbf{u} = \mathbf{u}(x) \) is independent of time. Since the scope of our work is restricted to stationary turbulence, we only give a brief overview of the justification of the time-dependent RANS equations. In order to include the transient term, there are two main, widely-used arguments. One of them makes use of the ergodicity hypothesis of turbulence and the other one uses multiple time scales \([66]\).

For more details on the ergodicity and multiple time scales approaches we refer the reader to \([30]\) and \([126]\), respectively. Here we only give an overview of the derivation with multiple time scales. The derivation starts by arguing that even though the definition given in (6.1) is mathematically well-defined for stationary turbulence, in practice instead of letting \( t \to \infty \), one can select a fixed averaging time \( t_a \) that is long enough for “smoothing out” the turbulent fluctuations. Then, the average pressure at time \( t \), for example, would be defined as

\[ p(x,t) = \frac{1}{t_a} \int_{t-a/2}^{t+a/2} p(x,\theta) d\theta. \tag{6.21} \]

The basic idea behind the derivation is to assume that the flow is characterized by two different time scales. One fast time scale \( t_1 \) describing the rapid fluctuations, and a slow time scale \( t_2 \) describing the large-scale fluid motion. Given these conditions, the averaging time \( t_a \) should be selected in such a way that \( t_1 \ll t_a \ll t_2 \). In this way, the turbulent fluctuations will be averaged out, while keeping the slow time scale behavior.

The previous argument is now widely accepted, however, it does not take into account the fact that turbulent flows have more than two time scales. This is mentioned by Wilcox himself \([126]\). In any case, for practical purposes, the time-dependent RANS
6.1 Turbulence Modeling

Equations are widely used in the following form

\[
\nabla \cdot \vec{\tau} = 0, \hspace{1cm} (6.22a)
\]

\[
\rho \left( \partial_t \vec{u} + \vec{u} \cdot \nabla \vec{u} \right) = -\nabla p + \mu \nabla^2 \vec{u} + \nabla \cdot \vec{R}. \hspace{1cm} (6.22b)
\]

6.1.5 Turbulence Closure Models

In the present section, we introduce the turbulence models which we will use with the practical purpose of computing the losses of mechanical energy for isothermal turbulent flows. The fundamental problem of turbulence modeling is to describe the Reynolds stress tensor \( \vec{R} \). The RANS models are typically classified in terms of the number of additional PDEs which one must solve on top of the original Navier-Stokes equations. There are zero-equation, one-equation and two-equation models. The zero-equation models are also commonly called algebraic, this because they involve no additional differential equation.

The \( k-\varepsilon \) model is an example of a two-equation model because it introduces two new partial differential equations. One modeling the kinetic energy of turbulence \( k \), and another one describing the dissipation rate \( \varepsilon \). Now we will discuss the Boussinesq turbulent-viscosity assumption, which forms the basis for the turbulence models that we will use.

6.1.6 Boussinesq Turbulent-Viscosity Assumption

Many turbulence models, and in particular the two-equation \( k-\varepsilon \) model and the algebraic LVEL model, make use of the Boussinesq turbulent-viscosity approximation. The idea of this approximation is to provide a Newtonian type of closure for the Reynolds stresses, by relating them to the mean flow strain rate. In the most general case, the Boussinesq assumption reads [43]

\[
\vec{R} := -\rho \vec{u}' \vec{u}' = \mu_t \left( \nabla \vec{u} + (\nabla \vec{u})^T - \frac{2}{3} (\nabla \cdot \vec{u}) \mathbf{I} \right) - \frac{2}{3} \rho k \mathbf{I}, \hspace{1cm} (6.23)
\]

where \( \mu_t \) [Pas] is the turbulent (dynamic) viscosity and \( k \) [J kg\(^{-1}\)] is the specific turbulent kinetic energy. Since \( \text{tr} \left( \nabla \vec{u} + (\nabla \vec{u})^T - \frac{2}{3} (\nabla \cdot \vec{u}) \mathbf{I} \right) = 0 \), we add the last term in (6.23), in order to ensure that the trace of \( \vec{R} \) equals \(-2\rho k\), according to (6.18). In the particular case of incompressible flows, the second term in (6.23) vanishes. In addition, it is customary to include the turbulent kinetic energy term in the normal stresses by defining \( p_t := \overline{p} + \frac{\rho k}{2} \) [Pa] as the new mean turbulent pressure [43]. After doing this and substituting...
 Isothermal Turbulent Flow

(6.23) in (6.22), the RANS equations take the form

\[ \nabla \cdot \mathbf{u} = 0, \quad (6.24a) \]
\[ \rho \left( \partial_t \mathbf{u} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p_t + (\mu + \mu_t) \nabla^2 \mathbf{u}. \quad (6.24b) \]

Sometimes it is convenient to rewrite the RANS momentum equations (6.24b), in terms of an effective viscosity \( \mu_e \) [Pa⋅s], which is defined by \( \mu_e := \mu + \mu_t \).

We would like to point out that the turbulent viscosity \( \mu_t \) is in general not constant. In fact, (6.23) is the definition for \( \mu_t \) and hence, the turbulent viscosity \( \mu_t \), is a property of the flow rather than a property of the fluid [66, 126]. The main difference between the models which we will present below, is the way in which \( \mu_t \) is modeled.

### 6.2 The \( k-\varepsilon \) Model

In this section we introduce the \( k-\varepsilon \) model which is one of the most used turbulence models in industry. The \( k-\varepsilon \) model introduces two additional transport equations that describe the specific turbulent kinetic energy \( k [J kg^{-1}] \) (defined in (6.19)) and the specific dissipation rate of turbulent kinetic energy \( \varepsilon [J kg^{-1} s^{-1}] \), which is defined as [86]

\[ \varepsilon := \frac{\mu}{2\rho} \left\| \nabla \mathbf{u}' + (\nabla \mathbf{u}')^\top \right\|^2, \quad (6.25) \]

where \( \| \cdot \| \) denotes the Frobenius norm of a tensor (see Appendix A, (A.6)). The turbulent viscosity \( \mu_t \) is given by the Kolmogorov-Prandtl relation

\[ \mu_t = C_\mu \frac{\rho k^2}{\varepsilon}, \quad (6.26) \]

where \( C_\mu \) is an empirical model constant. The previous relation is obtained on the basis of dimensional arguments, we refer the reader to [43] for more details. Relation (6.26) closes the RANS continuity and momentum equations (6.24). However, one still needs transport equations which model \( k \) and \( \varepsilon \).
6.2 The $k$-$\varepsilon$ Model

6.2.1 Transport Equations for $k$ and $\varepsilon$

The two additional transport equations modeling the specific turbulent kinetic energy $k$ and the specific dissipation rate of turbulent energy $\varepsilon$ are

$$
\rho \left( \partial_t k + \overline{u^i} \nabla k \right) = \nabla \cdot \left( \left( \mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right) + \mu_t \nabla \overline{u^i} : \left( \nabla \overline{u^i} + \left( \nabla \overline{u^i} \right)^T \right) - \rho \varepsilon, \quad (6.27)
$$

$$
\rho \left( \partial_t \varepsilon + \overline{u^i} \nabla \varepsilon \right) = \nabla \cdot \left( \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right) + C_{\varepsilon 1} \frac{\varepsilon \mu_t}{k} \nabla \overline{u^i} : \left( \nabla \overline{u^i} + \left( \nabla \overline{u^i} \right)^T \right) - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k}, \quad (6.28)
$$

where $\sigma_k$, $\sigma_\varepsilon$, $C_{\varepsilon 1}$ and $C_{\varepsilon 2}$ are dimensionless empirical constants. The derivation of these equation is out of the scope of this thesis and for further information, we refer the reader to [17,126].

The left hand side of (6.27) represents the rate of change of turbulent kinetic energy $k$. The first term on the right hand side represents the rate of change of turbulent kinetic energy $k$ by molecular and turbulent diffusion. The second term represents the rate of production of turbulent kinetic energy $k$ by the action of mean shear. The last term represents the rate of viscous dissipation of turbulent kinetic energy (also called decay) [57].

The last term in (6.27) involves the specific dissipation rate of turbulent kinetic energy $\varepsilon$, which in turn is modeled by equation (6.28). The first term on the right hand side of (6.28) represents the molecular and turbulent diffusion of $\varepsilon$. The second term represents the production of $\varepsilon$ due to the action of mean shear and the last term represents the destruction of $\varepsilon$.

It is important to mention that equation (6.27) can be obtained analytically by first deriving equations for the Reynolds stress tensor $\mathbf{R}$ from (6.21) and then taking the trace of this tensor in order to arrive at an equation for $k$. Equation (6.27) is finally obtained by making use of the Boussinesq assumption (6.23) and it does not require additional modeling [126]. In contrast to this, each of the right hand side terms in (6.28) are actually additional models for the diffusion, production and destruction terms of $\varepsilon$. These models are obtained based on dimensional analysis arguments [17]. In fact, it is possible to obtain an analytical equation for $\varepsilon$. However, this equation involves time averages of products of up to three fluctuating velocity components. These kind of correlations are essentially impossible to measure (6.28). A way to overcome this problem is to use the empirical equation (6.28) to model $\varepsilon$.

The dimensionless constants involved in the $k$-$\varepsilon$ model, are chosen in such a way that the model reproduces certain properties of isotropic turbulence, such as the decay in time of homogeneous turbulence, the measurements in shear layers in local equilibrium.
and the logarithmic law of the wall in boundary layers \cite{17, 86}. In our particular case, we use the implementation of the $k-\varepsilon$ model available in the commercial FEM code COMSOL Multiphysics \cite{21}. The values of the constants used in the code are

$$C_\mu = 0.09, \quad C_{\varepsilon 1} = 0.1256, \quad C_{\varepsilon 2} = 1.92, \quad \sigma_k = 1, \quad \sigma_\varepsilon = 1.3.$$ \hspace{1cm} (6.29)

### 6.2.2 Wall-Functions

In order to apply the $k-\varepsilon$ model to wall-bounded flows, we need to provide boundary conditions for the velocity and the two turbulence variables $k$ and $\varepsilon$. The $k-\varepsilon$ model is suitable for modeling isotropic turbulence. However, in regions close to a solid wall, turbulence is not isotropic and it exhibits deviations which vary in magnitude and direction. Therefore, (6.27) and (6.28) do not hold close to solid walls, and we need additional modeling \cite{17, 86}. The expressions for the average velocity profile in a turbulent flow are based on both analysis and measurements. They are semi-empirical and require constants which are determined from experimental data.

There are basically two main alternatives for modeling turbulent flow close to solid walls. The first alternative for modeling solid walls, is to modify the equations and add terms in order to take into account the near-wall effects. Such an approach, however, requires very fine meshes near the wall in order to resolve the very steep gradients near the wall and therefore it is restricted to moderate Reynolds numbers. The second alternative, which is the one we adopt here, is to make use of the so-called laws of the wall. These are empirical relations which link the average velocity and the wall friction. We will refer to two different laws, the law of the wall and the logarithmic law of the wall. These laws hold in different regions of the flow (see Figure 6.1), the law of the wall next to the wall where the velocity profile is linearly dependent on the distance to the wall, while the logarithmic law of the wall hold in a region where the velocity depends on the logarithm of the distance to the wall. In particular, we will use the logarithmic law of the wall in order to derive boundary conditions for the $k-\varepsilon$ model at certain distance from the physical wall, hence avoiding the need to resolve the boundary layer near the wall.

Before proceeding it is instructive to discuss some of the differences between laminar and turbulent velocity profiles between two flat plates. In such a situation, the flow can be described by one single average velocity component $u_t$, parallel to the wall. Since the velocity is parallel to the wall, we can conclude, using continuity, that $u_t$ depends only on the perpendicular distance to the wall, which we will denote by $y$. Such a description is valid for both laminar and turbulent flows. Figures 6.1(a) and 6.1(b) show the fully developed velocity profiles in the case of laminar and turbulent flow, respectively. In the case of laminar flow, we observe the well-known parabolic profile. In the turbulent case, the eddy motions and hence the turbulent viscosity $\mu_t$ are much larger than their
molecular counterpart in the core region of a turbulent flow. The eddy motion reduces in intensity close to the wall until it becomes 0 at the wall due to the no-slip condition \( u' = 0 \) and \( u_t = 0 \). Consequently, the velocity profile in a turbulent flow changes slowly in the core of a turbulent flow and it is very steep in a thin layer close to the wall [15].

From Figure 6.1(b), we can expect at least two length scales associated with a turbulent flow. One of them corresponds to the thin region close to the wall in which the velocity profile changes rapidly, and a second one is related to the nearly uniform velocity profile in the so-called outer region or turbulent layer of the flow. The region very close to the wall shows a nearly linear velocity profile, this inner layer is called the viscous sublayer. The so-called outer region (also called core region) shows a nearly constant velocity. Between these two regions, we identify yet two more layers, namely the logarithmic and buffer layers. In the logarithmic layer, the velocity can be expressed as the logarithm of the distance to wall, this relation is referred to as the logarithmic law of the wall. The other region, called the buffer layer, is located between the viscous sublayer and the logarithmic layer. In this region, both the law of the wall and the logarithmic law of the wall, do not hold. These regions are schematically presented in Figure 6.1(b).

The logarithmic law of the wall is derived via the method of matched asymptotic expansions. However, here we will present the heuristic derivation given in [66,119]. For a detailed derivation we refer the reader to [126], here we only provide an overview.

Under the assumption that the flow close to the wall is not affected by the flow far from the wall, one should expect to find all the relevant scales with flow parameters close to the wall. We will use the wall shear stress \( \tau_w \), the viscosity \( \mu \) and the density \( \rho \). The wall shear stress is defined as

\[
\tau_w := \mu \left. \partial_y u_t \right|_w,
\]

Figure 6.1: Fully developed velocity profiles between two flat plates. Figure 6.1(a) shows the well-known parabolic profile for laminar flows and Figure 6.1(b) shows the turbulent average velocity profile. The outer, logarithmic, buffer and viscous layer are schematically shown.
where the subindex \( w \) means that the derivative is evaluated at the wall of the pipe. Based on these quantities we can introduce a characteristic velocity in the viscous sublayer. This is the so-called \textit{friction velocity} \( u_{\tau} \), which is defined as

\[
u = \sqrt{\frac{\tau}{\rho}}, \tag{6.31}
u = \frac{\nu}{\rho}, \tag{6.32}
\]

We now use the friction velocity to nondimensionalize the velocity \( u_t \) and the perpendicular distance to the wall \( y \). We do this by defining

\[
u^{+} := \frac{\nu u_{\tau}}{\nu}, \quad u^{+} := \frac{u_t}{u_{\tau}}, \tag{6.32}
u^{+} := \frac{\nu u_{\tau}}{\nu}, \quad u^{+} := \frac{u_t}{u_{\tau}}, \tag{6.32}
\]

where \( u^{+} \) and \( y^{+} \) are the dimensionless velocity and perpendicular distance to the wall, respectively. Here \( \nu = \mu/\rho \) denotes the kinematic viscosity and the superscript \( ^{+} \) signals that the quantities are dimensionless. These dimensionless variables are commonly referred to as wall units. We note that \( y = 0 \) and \( y^{+} = 0 \) now represent the location of the wall (see Figure 6.1).

In the viscous sublayer, the wall damps the eddy motions and thus the flow is essen-
6.2 The $k$-$\varepsilon$ Model

The velocity profile in this region is linear and in terms of the dimensionless variables is given by the simple relation

$$u^+ = y^+,$$  \hspace{1cm} (6.33)

which is known as the law of the wall. This expression has been confirmed experimentally and it is found to correctly describe the turbulent flow profile over smooth flat surfaces in the region $0 \leq y^+ \leq 5$.

The velocity in the viscous sublayer and in the outer layer are matched in an intermediate region which is referred to as the logarithmic layer (also called overlap layer), where the logarithmic law of the wall holds. In terms of the dimensionless variables $u^+$ and $y^+$ it reads

$$u^+ = \frac{1}{K} \ln y^+ + C,$$  \hspace{1cm} (6.34)

where $K$ denotes the von Kármán constant and $C$ is an additive constant. Both of these constants are determined experimentally. For the simulations presented here (for the $k$-$\varepsilon$ model only) we used $K = 0.41$ and $C = 5$ [21, 126]. The law of the wall and the logarithmic law of the wall which we presented here were derived for flow between two plates, however, these expressions are also valid for pipe flow. Figure 6.2 for example, shows a comparison between the velocity profiles given by the law of the wall (6.33), the logarithmic law of the wall (6.34) and experimental data obtained by McKeon et.al. for turbulent flow in a pipe [67]. The logarithmic law of the wall is accurate in the overlap layer and reasonably accurate in the outer layer. For this particular experiment, the logarithmic law of the wall works very well in the region $20 < y^+ < 200$, however it is commonly recommended to apply the law of the wall in the region $30 < y^+ < 100$ [15].

6.2.3 Boundary Conditions for the $k$-$\varepsilon$ Model

The logarithmic law of the wall (6.34) can be used for prescribing boundary conditions for the $k$-$\varepsilon$ model at a certain distance from the physical wall, i.e., at $y = y_p$, where $y_p$ satisfies $y_p^+ := y_p u_+ / \nu \in [30, 100]$ (such that (6.34) remains valid, see Figure 6.2). In order to do this, we need to express $u_+$ in terms of the original field variables at a distance $y_p$ from the wall.

In what follows we will show that at $y = y_p$ we have $u_+ = C^{1/4} \varepsilon^{1/2}$. We will do this by combining two alternative expressions for $\varepsilon$. We will only present a schematic derivation and for further details, we refer to the reader to [17, 87].

First, since $y_p$ is selected such that $y_p^+ \geq 30$, the laminar part of the shear stress is negligible and therefore we can write the shear stress as $\tau = \mu \partial_x u_+$. Substituting the previ-
ous expression for the shear stress into the definition of the friction velocity \( u_\tau := \sqrt{\tau / \rho} \), we obtain

\[
u_\tau^2 = \frac{\mu_t}{\rho} \partial_y u_t.
\] (6.35)

The derivative \( \partial_y u_t \) can be computed by differentiating the logarithmic law of the wall (6.34) with respect to \( y \). This yields

\[
\partial_y u_t = \frac{u_\tau}{K_y}.
\] (6.36)

Substituting (6.36) into (6.35) and replacing the turbulent viscosity \( \mu_t \) with the Kolmogorov-Prandtl relation (6.26), we obtain a first expression for \( \varepsilon \)

\[
\varepsilon = \frac{C\mu k^2}{u_\tau K_y}.\] (6.37)

We can obtain an alternative expression for \( \varepsilon \), by considering an equilibrium state, which means that the production of turbulent kinetic energy (second right hand side term in (6.27)) balances the dissipation of turbulent kinetic energy \( \rho \varepsilon \) (third r.h.s. term in (6.27)). Under this assumption \[17\], (6.27) takes the form

\[
\mu_t \nabla \mathbf{u} : \left( \nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) - \rho \varepsilon = 0.
\] (6.38)

Since we are considering the case of parallel flow, the previous equation simplifies to

\[
\mu_t \left( \partial_y u_t \right)^2 - \rho \varepsilon = 0.
\]

Solving for \( \varepsilon \) in this equation we obtain

\[
\varepsilon = \frac{(\mu_t / \rho) (\partial_y u_t)^2}{16}.
\]

Substituting (6.26) and (6.36) into this equation, and solving for \( \varepsilon \) we arrive at a second alternative expression for \( \varepsilon \), namely

\[
\varepsilon = \frac{C_1^{1/2} k^{1/2}}{u_\tau K_y}.
\] (6.39)

The right hand sides of the two alternative expression for \( \varepsilon \), namely (6.37) and (6.39), can be equalized in order to eliminate \( \varepsilon \) from the equations. Doing this, and solving for \( u_\tau \) we finally obtain that in the logarithmic layer (i.e., at \( y = y_p \)),

\[
u_\tau = C_1^{1/4} k^{1/2}.
\] (6.40)

The previous expression is used in several software packages which support turbulent flow simulation with two-equation models. In our particular case, we will present the variant which is implemented in the code COMSOL Multiphysics \[22,87\]. This is obtained by rewriting the logarithmic law of the wall (6.34) in dimensional variables and
multiplying it by $u_\tau^2$, which yields

$$u_\tau u_\tau = u_\tau^2 \left( \frac{1}{K} \ln y^+ + C \right). \quad (6.41)$$

Replacing $u_\tau^2$ by $\tau/\rho$ (by definition of $u_\tau$), and $u_\tau$ by $C_\mu^{1/4}k^{1/2}$ (see (6.40)) we obtain.

$$u_\tau C_\mu^{1/4}k^{1/2} = \frac{\tau}{\rho} \left( \frac{1}{K} \ln y^+ + C \right). \quad (6.42)$$

Solving for $\tau$ in the previous equation we obtain the following expression for the wall shear stress

$$\tau = \frac{\rho C_\mu^{1/4}k^{1/2}}{\frac{1}{K} \ln y^+ + C} u_\tau. \quad (6.43)$$

Evaluating the previous relation at $y = y_p$ provides us with the boundary condition for the shear stress at a certain distance from the physical wall. It is important to mention that (6.43) was derived under the assumption of parallel flow, i.e., for straight channel or pipe flow. In this setting the normal stress at $y = y_p$ is described by one single scalar which is given by $\tau$ in (6.43). However, in the case of a curved boundary or of a non-parallel flow, the normal stress is a vector (see (2.43)). A practical solution is to replace $u_\tau$ by $\mathbf{u}$ in (6.43) and take this expression as an approximation for the normal stress $\mathbf{T}_n$. Such an approach is expected to work correctly for regions in which the flow is attached and the turbulence is fully developed in the boundary layer [86]. The boundary condition which we obtain reads

$$\mathbf{T}_n = (\mu + \mu_t) \left( \nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) \mathbf{n} = \frac{\rho C_\mu^{1/4}k^{1/2}}{\frac{1}{K} \ln y^+ + C} \mathbf{u}, \quad \text{at} \quad y = y_p, \quad (6.44)$$

where $y_p^+ = u_\tau y_p/\nu$. For the turbulent kinetic energy, one considers that there is no flux of turbulent kinetic energy between the near wall region and the core region of the flow, i.e., $\mathbf{n} \cdot \nabla k = 0$. The boundary condition for the dissipation $\varepsilon$ can obtain by combining (6.37) and (6.40), which yields $\varepsilon = C_\mu^{3/4}k^{3/2}/K y_p$.

Summarizing, the boundary conditions for the $k-\varepsilon$ model at a solid wall are not prescribed at the physical wall of the pipe ($y = 0$), but rather at a certain distance from the physical wall, i.e., at $y = y_p$, where $y_p$ is chosen such that $y_p^+ := y_p u_\tau/\nu \in [30, 100]$. The
boundary conditions read
\[
(\mu + \mu_t) \left( \nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) \mathbf{n} = \frac{\rho C_{\mu}^{1/4} k^{1/2}}{K} \ln y_p^+ + C, \\
\mathbf{n} \cdot \nabla k = 0, \\
\epsilon = \frac{C_{\mu}^{1/4} k^{3/2}}{K y}, \\
\text{at } y = y_p,
\]
where \( y_p^+ = u^+ y_p / \nu \).

6.3 The LVEL Model

In this section we introduce the LVEL turbulence model. The model was named LVEL to emphasize the fact that the model only requires the distance from the wall (referring to the first L in the name) and the local velocity (referring to the VEL characters in the name), in order to compute the turbulent viscosity \( \mu_t \). The expression for the turbulent viscosity used in the LVEL model is derived from Spalding’s law of the wall.

6.3.1 Spalding’s Law of The Wall

Spalding’s law of the wall is an empirical expression for fully developed turbulent flow between plates (which also holds for pipe flow). In contrast to the logarithmic law of the wall (6.34), Spalding’s law of the wall provides a good fit not only in the logarithmic layer, but in the viscous sublayer and in the buffer layer as well. Spalding obtained this expression in the following way [3].

First, from the logarithmic law of the wall (6.34) he expressed \( y^+ \) as a function of \( u^+ \), as follows
\[
y^+ = \frac{1}{\mathcal{E}} e^{\mathcal{K}u^+}, \tag{6.46}
\]
where \( \mathcal{K} \) is the von Karman constant and \( \mathcal{E} \) is another constant given by \( \mathcal{E} = \exp (\mathcal{K} C) \), where \( C \) is the constant from the logarithmic law of the wall (6.34).

Then he modified the right hand side of (6.46) by adding the laminar term \( u^+ \) (see (6.33)) and subtracting some terms from the Taylor expansion of the exponential function. The
expression thus obtained is the so-called Spaldings law of the wall, and it reads [111]

\[ y^+ = u^+ + \frac{1}{C} \left( e^{Ku^+} - 1 - Ku^+ - \frac{(Ku^+)^2}{2} - \frac{(Ku^+)^3}{6} - \frac{(Ku^+)^4}{24} \right). \]  (6.47)

In the original paper, the constants were taken as \( K = 0.4 \) and \( C = 5.5 \) [111]. Figure 6.3 shows a comparison between Spalding’s law of the wall and the experimental data obtained of McKeon et.al. [67] and of Reichardt et.al. [98]. Spalding’s law of the wall correctly represents the linear and logarithmic behavior of the velocity profile in the viscous and logarithmic layers, respectively. The buffer layer is also nicely captured by this expression and hence, this law provides a good approximation everywhere in the domain.

### 6.3.2 Turbulent Viscosity and Local Reynolds Number

One of the utilities of Spalding’s law of the wall is that it is possible to derive an analytical expression for the turbulent viscosity \( \mu_t \). This can be done in the following way. On
the basis of the RANS equations (see (6.24)), for parallel flow we have

$$\tau_w = \mu_e \frac{dy^+}{du^+},$$  \hspace{1cm} (6.48)

where $\mu_e$ is the effective viscosity, $\mu_e = \mu + \mu_t$ and where the dimensionless distance $y^+$ and velocity $u^+$ have the form $u^+ = u_t/u_\tau$, $y^+ = y u_\tau/\nu$. Applying the chain rule to these expressions, we can calculate $\frac{du_t}{dy}$, the expression reads

$$\frac{du_t}{dy} = \frac{u_t^2}{\nu} \frac{du^+}{dy^+}.$$  \hspace{1cm} (6.49)

Substituting the definition (6.31) for $u_\tau$ into (6.49) and combining it with (6.48) we obtain

$$\frac{1}{\mu_e} = \frac{1}{\mu} \frac{du^+}{dy^+}.$$  \hspace{1cm} (6.50)

This provides us with a way of computing the effective viscosity $\mu_e$ once the velocity profile is known. In the particular case of Spalding’s law of the wall we have

$$\mu_e = \frac{dy^+}{du^+} = 1 + \frac{\kappa}{\epsilon} \left( e^{\kappa u^+} - 1 - \kappa u^+ - \frac{(\kappa u^+)^2}{2} - \frac{(\kappa u^+)^3}{6} \right).$$  \hspace{1cm} (6.51)

The previous expression allows us to compute the effective viscosity as function of the dimensionless velocity field $u^+$. By construction, (6.51) should yield accurate results in the case of straight channels and pipes. In the case of curved surfaces, the flow is no longer parallel to the surface and (6.51) does not need to hold. However, we can still apply the same expression by replacing the parallel velocity component $u_t$ by the magnitude of the velocity $|\bar{u}|$ and the perpendicular distance to the wall $y$ by the nearest distance to the wall $\delta[m]$. The nearest distance to the wall $\delta[m]$ is defined by

$$\delta(x) := \min_{x_0 \in \Gamma} |x_0 - x|,$$  \hspace{1cm} (6.52)

where $\Gamma$ is the (possibly curved) wall of the pipe.

In order to be able to use (6.51) for solving the RANS system of equations (6.24), we still need to relate $u^+$ to the average velocity field $\bar{u}$. This can be done by considering the local Reynolds number $Re_{loc}(x)$, which is defined as

$$Re_{loc}(x) := \frac{\rho |\bar{u}| \delta}{\mu},$$  \hspace{1cm} (6.53)

where $\rho$ is the density of the fluid, $\bar{u}$ the average velocity and $\mu$ the (laminar) viscosity of the fluid. Alternatively, we can express the local Reynolds number $Re_{loc}$ in terms of
The dimensionless velocity $u^+$. In fact we have,

$$Re_{loc}(x) = \frac{|u|_x \delta u_x \beta}{\mu} = u^+ y^+. \quad (6.54)$$

By substituting Spalding’s law of the wall (6.47) in the equation above, we obtain an approximate expression for the local Reynolds number in terms of $u^+$, which we will denote by $Re_{est}$. The approximate expression obtained in this way reads

$$Re_{est}(u^+) := u^+ \left[ u^+ + \frac{1}{\varepsilon} \left( e^{Ku^+} - 1 - K u^+ - \frac{(Ku^+)^2}{2} - \frac{(Ku^+)^3}{6} - \frac{(Ku^+)^4}{24} \right) \right]. \quad (6.55)$$

Equations (6.55) and (6.51) form the basis for the numerical solution of the RANS system of equations (6.24).

### 6.3.3 Solution Algorithm

Now we will discuss how (6.51) and (6.55) can be used for solving the RANS system of equations (6.24). The numerical solution of this system of equations, requires the effective viscosity $\mu_e$ as input. The effective viscosity can be computed from (6.51), however, this expression involves $u^+$, which in turn is dependent on $\tau$. Therefore an iterative procedure is required.

The dimensionless velocity $u^+$ is related to $\tau$ via the equation (6.32), and in principle it is possible to construct an iterative procedure based on this relation. However, the computation of the friction velocity $u_\tau$ becomes cumbersome in the case of curved boundaries. A way to circumvent this problem is to recur to the local Reynolds number $Re_{loc}$.

The local Reynolds number $Re_{loc}$ plays an important role in the iterative solution method for the LVEL model. The computation of $Re_{loc}$ requires as input the nearest distance to the wall $\delta$. The wall distance $\delta$ can be computed using search-based geometrical procedures, however, these procedures can become prohibitively expensive when dealing with complex geometries. For complex geometries, it is more efficient to use an alternative method based on the solution of a Poisson equation. This method for computing the wall distance was an original key component of the LVEL method [3]. This idea was originally developed by Spalding [110] and later refined by Tucker [123] and other researchers. Eventually, this method for computing wall distances found its way into most of the commercial CFD codes [3]. For more details on this topic we refer the reader to [122, 123]. The computation of the wall distance $\delta$, is performed as a preprocessing step of the solution method.
The solution method which we will use for solving the RANS system of equations (6.24) works in an iterative manner with respect to the dimensionless velocity \( u^+ \). The structure of the algorithm is the following.

**Solution Algorithm for the LVEL Model**

**INPUT:** Geometry \( \Omega \), boundary conditions for the RANS system (6.24) and tolerance \( TOL \)

**OUTPUT:** Velocity \( \overline{u} \) and pressure \( p_t \)

1. Compute the wall distance \( \delta \).
2. Set counter to \( i = 0 \) and set \( CORRECTION = 1 \).
3. Provide an initial guess for the dimensionless velocity, i.e., \( u^{+(i)}(x) = 0 \).

4. while \( CORRECTION > TOL \) do
   (a) Evaluate \( \mu_e(u^{+(i)}) \) from (6.51), compute \( \text{Re}_{\text{est}}(u^{+(i)}) \) and \( \frac{d\text{Re}_{\text{est}}}{du^+}(u^{+(i)}) \), according to (6.55).
   (b) Solve the RANS equations (6.24) with the input boundary conditions and find \( \overline{u} \) and \( p_t \).
   (c) Use the current velocity \( \overline{u} \) to compute \( \text{Re}_{\text{loc}} \) according to the formula (6.53).
   (d) Compute the next iteration of the dimensionless velocity in the following way
   \[
   u^{+(i+1)} = u^{+(i)} + \frac{\text{Re}_{\text{loc}} - \text{Re}_{\text{est}}(u^{+(i)})}{\frac{d\text{Re}_{\text{est}}}{du^+}(u^{+(i)})}.
   \]
   (e) Calculate correction
   \[
   CORRECTION = \left\| u^{+(i+1)} - u^{+(i)} \right\|_\infty.
   \]
   (f) Increase counter \( i = i + 1 \).
5. end while
6. Return the converged velocity \( \overline{u} \) and the pressure \( p_t \).

The algorithm above, starts by setting \( u^{+(0)} \equiv 0 \). This step simply means that we set the turbulent viscosity to 0, which in turn implies \( \mu_e = \mu \). After computing the average velocity \( \overline{u} \) and the pressure \( p_t \), we update the dimensionless velocity \( u^+ \). The computation of the new dimensionless velocity in step 4d, uses a Newton-like iteration based on the estimate of the local Reynolds number, given in (6.55). The new value of \( u^+ \) is used
6.4 Setup for Periodic Corrugated Pipes

6.4 Setup for Periodic Corrugated Pipes

The turbulence models which we discussed above can be readily applied to arbitrary geometries. In our particular case we will apply them to simulate turbulent flow through periodic corrugated pipes. We will assume axially symmetric flow.

In Section 2.5.4, we introduced the notion of a periodic corrugated pipe for axially symmetric geometries. The periodicity condition reads \( R_w(x) = R_w(x + L) \), where \( R_w(x) \) denotes the radius of the pipe at the axial position \( x \) and \( L \) is the period of the pipe. For the sake of clarity, in Figure 6.4 we show a diagram of a typical corrugated pipe. Due to the assumption of axial symmetry, we can consider only one azimuthal section of the pipe (for instance \( \phi = 0 \)). The domain \( \Omega \) corresponds to a single period of the pipe, i.e.,

\[
\Omega := \left\{ r e_r + xe_x \in \mathbb{R}^2 \mid 0 \leq x \leq L, 0 \leq r \leq R_w(x) \right\}.
\] (6.56)

The boundary of the domain is decomposed as \( \partial \Omega = \Gamma_{in} \cup \Gamma_{out} \cup \Gamma \), where \( \Gamma_{in}, \Gamma_{out} \) and \( \Gamma \) stand for the inlet, outlet and wall of the pipe, respectively, i.e.,

\[
\Gamma_{in} := \left\{ r e_r \in \mathbb{R}^2 \mid 0 \leq r \leq R_w(0) \right\}, \tag{6.57a}
\]
\[
\Gamma_{out} := \left\{ r e_r + Le_x \in \mathbb{R}^2 \mid 0 \leq r \leq R_w(0) \right\}, \tag{6.57b}
\]
\[
\Gamma := \left\{ R_w(x)e_r + xe_x \in \mathbb{R}^2 \mid 0 \leq x \leq L \right\}. \tag{6.57c}
\]
6.4.1 Periodicity Decomposition

In Section 3.3, we presented a numerical method for simulating fully developed laminar flow through corrugated pipes. The idea of the method was to reduce the computational domain to just one single period by decomposing the pressure as the sum of a mean axial gradient and an oscillating part due to the wall-shape. The same technique was used in Section 4.3, for solving the problem of forced convection with constant heat flux.

In the case of fully developed turbulent flow through corrugated pipes, we can also apply the same technique. In analogy with the laminar case, we decompose the pressure as follows

\[ p_t = \tilde{p}_t - \beta x, \]

where \( \tilde{p}_t \) [Pa] represents the pressure deviations due to wall-shape and \( \beta \) [Pa m\(^{-1}\)] is the average pressure gradient in the axial direction. The advantage is that \( \tilde{p}_t \) is periodic and hence we can reduce the computational domain to just one period. The average pressure gradient then appears as the volume force \( \beta e_x \) in the momentum equations.

We remark that the pressure decomposition given in (6.58) can be applied to the LVEL model as well as to the \( k-\varepsilon \) model, however, the system of equations and the boundary conditions are slightly different in each case. This is no surprise because the \( k-\varepsilon \) model involves more variables than the LVEL model.
6.4.2 The $k$-$\varepsilon$ System of Equations

In the case of the $k$-$\varepsilon$ model, the system of equations which we need to solve in the domain $\Omega$ reads

\begin{align*}
\nabla \cdot \mathbf{u} &= 0, \\
 \rho (\partial_t \mathbf{u} + \mathbf{u} \cdot \nabla \mathbf{u}) &= -\nabla \tilde{p}_t + (\mu + \mu_t) \nabla^2 \mathbf{u} + \beta \varepsilon, \\
 \rho (\partial_t k + \mathbf{u} \cdot \nabla k) &= \nabla \cdot \left( \left( \mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right) + \mu_t \nabla \cdot \left( \nabla \mathbf{u} + \left( \nabla \mathbf{u} \right)^T \right) - \rho \varepsilon, \\
 \rho (\partial_t \varepsilon + \mathbf{u} \cdot \nabla \varepsilon) &= \nabla \cdot \left( \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right) + C_{\varepsilon 1} \frac{\varepsilon \mu_t}{k} \nabla \cdot \left( \nabla \mathbf{u} + \left( \nabla \mathbf{u} \right)^T \right) - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k},
\end{align*}

where $\tilde{p}_t$ is the pressure fluctuation defined in (6.58) and where the turbulent viscosity is given by $\mu_t = C_\mu \rho k^2/\varepsilon$. The system of equations is subject to the following set of periodic boundary conditions

\begin{align*}
\tilde{p}_t (r, 0) &= \tilde{p}_t (r, L), & \mathbf{u} (r, 0) &= \mathbf{u} (r, L), \\
k (r, 0) &= k (r, L), & \varepsilon (r, 0) &= \varepsilon (r, L).
\end{align*}

The wall boundary conditions read (see Section 6.2.3)

\begin{align*}
\left( \mu + \mu_t \right) \left( \nabla \mathbf{u} + \left( \nabla \mathbf{u} \right)^T \right) \cdot \mathbf{n} &= \frac{C_{\mu}^{1/4} \bar{y}_p^{1/2}}{K \ln \bar{y}_p} + C \mathbf{u}, \\
\mathbf{n} \cdot \nabla k &= 0, \\
\varepsilon &= \frac{C_{\mu}^{3/4} \bar{y}_p^{3/2}}{K \bar{y}_p} \text{ at } \Gamma_{y_p}
\end{align*}

where $\Gamma_{y_p}$ represents the curve which is at distance $y_p$ from the physical wall $\Gamma$, i.e.,

\[ \Gamma_{y_p} := \{ x \in \Omega | \delta(x) = y_p \}. \]

6.4.3 The LVEL System of Equations

The system of equations which we need to solve in the case of the LVEL model is the following

\begin{align*}
\nabla \cdot \mathbf{u} &= 0, \\
 \rho (\partial_t \mathbf{u} + \mathbf{u} \cdot \nabla \mathbf{u}) &= -\nabla \tilde{p}_t + (\mu + \mu_t) \nabla^2 \mathbf{u} + \beta \varepsilon.
\end{align*}
The system is provided with the following boundary conditions

\begin{align}
\text{Periodicity:} & \quad \bar{p}_i (r, 0) = \bar{p}_i (r, L), \quad \bar{\varphi} (r, 0) = \bar{\varphi} (r, L), \quad (6.64a) \\
\text{No-slip:} & \quad \varphi (R_w(x), x) = 0 \quad \text{for} \quad 0 \leq x \leq L. \quad (6.64b)
\end{align}

The system of equations of the LVEL model has exactly the same form as the steady state Navier-Stokes equations which we used in the case of laminar flow. The difference comes from the turbulent viscosity \( \mu_t \). In the LVEL model, the turbulent viscosity is dependent on the velocity, i.e., \( \mu_t = \mu_t(\varphi) \). However, the relation is given in a complex manner via the equation (6.51). Due to the difficulty to find this function, the system of equations (6.63) needs to be solved with the algorithm described in Section 6.3.3.

For both models, we use a zeroth order continuation method with respect to the volume force \( \beta \), i.e., the solution for a given volume force is used as the initial guess for solving the problem with the next larger value of the pressure force.

### 6.5 Numerical Analysis

In the present section we will discuss the numerical implementation and results obtained with the \( k-\varepsilon \) and the LVEL turbulence models. We also provide validation of the model for turbulent flow in a straight pipe, by comparing our numerical results with the classical experiments by Nikuradse and with the more recent superpipe experiments by Zagarola et al. In the case of corrugated pipes, we provide comparison with DNS numerical simulations and with some experimental data as well. We conclude with a comparison between the \( k-\varepsilon \) model and the LVEL model.

#### 6.5.1 Discretization

The \( k-\varepsilon \) system of equations (6.59) and the LVEL system of equations (6.63) are solved with a mixed finite element model. In both turbulence models, the velocity \( \varphi \) is approximated with quadratic Lagrange elements and the pressure \( p_i \) is approximated with linear elements. In case of the \( k-\varepsilon \) model, the specific kinetic energy of turbulence \( k \) and the specific dissipation rate of turbulent energy \( \varepsilon \) are approximated with linear elements. Both turbulence models were implemented with the code Comsol Multiphysics [22]. For the \( k-\varepsilon \) model, we used the default implementation readily available in the code.

The \( k-\varepsilon \) discrete system of equations is solved with a segregated solver which works in an iterative way as follows. We start with an initial guess for \( \varphi, k \) and \( \varepsilon \), which we
use to compute the turbulent viscosity $\mu_t$. Next we solve the continuity (6.59a) and momentum equations (6.59b). Then we solve the equation for $k$ (6.59c) and subsequently the equation for $\varepsilon$ (6.59d). With the new values of $k$ and $\varepsilon$ we update $\mu_t$ and we repeat the procedure until convergence is reached. The non-linear flow equations are solved using Newton iteration while the transport equations are solved with a direct solver. The implementation of the algorithm works with a reformulation of the $k$-$\varepsilon$ equations in terms of $\ln k$ and $\ln \varepsilon$. This transformation guarantees that $k$ and $\varepsilon$ are positive and hence also $\mu_t$ remains positive [46].

In the case of the LVEL model, we implemented our own solution algorithm, which we presented in Section 6.3.3. In order to solve the system of equations (6.63), the LVEL solution algorithm also uses an iterative procedure. In contrast to the $k$-$\varepsilon$ model, the LVEL model involves no extra transport equations. Therefore we can expect less numerical efforts with the LVEL model than with the $k$-$\varepsilon$ model. At the same time, the number of iterations required to find a converged solution reduces.

### 6.5.2 Meshing

The type of meshes which can be applied to a specific problem, depend strongly on the characteristics of the geometry and on the nature of the equations to be solved.

For a **straight pipe**, we use a structured mesh with quadrilateral elements, as the one displayed in Figure 6.5. The distribution of elements in the axial direction is equidistant, while the distribution in the radial direction is done in such a way that the size of the elements decreases in a logarithmic fashion when approaching the wall. The logarithmic scaling of the elements in the radial direction is motivated by the law of the wall (see Section 6.2.2).

For corrugated pipes, we use a **boundary layer type mesh**, displayed as in Figure 6.6. The special feature of such a mesh is a layer of quadrilateral elements with logarithmic scaling next to the wall. In the rest of the regions, a triangular mesh is used. This type of mesh is advantageous for curved surfaces because the quadrilateral elements allow us to capture the boundary layer close to the wall, while the triangular mesh allows to describe complicated geometries while keeping the quality of the elements.

### 6.5.3 Mesh Independence

In order to obtain mesh independent solutions, the mesh was refined until a new refinement provided an insignificant variations in the volumetric flow rate $Q$. We note that an accurate value of $Q$ implies an accurate value of the resulting Reynolds number $Re$. 

Figure 6.5: Quadrilateral mesh for a straight pipe with 16 elements in the axial direction $x$, and 16 elements in the radial direction $r$.

Figure 6.6: Boundary layer mesh for a sinusoidal pipe with parameters $R_w(0), L = 9.33, L_e = 0.5$ and $a = 2.33$ (see Figure 3.14). The mesh has 241 quadrilateral elements and 210 triangular elements.

Figure 6.7(a) shows the relative variation in the computed volumetric flow rate versus the volume force term $\beta$ in the case of the $k$-$\varepsilon$ model. Figure 6.7(b) shows the variations for the LVEL model. We considered logarithmic quadrilateral meshes described as in Figure 6.5. The mesh sizes were $16 \times 16$, $32 \times 32$ and $48 \times 48$ and $64 \times 64$. A horizontal line signaling a relative error of 1% was added to Figures 6.7 for reference. In the case of the $k$-$\varepsilon$ model, the relative variation is below 1% for $\beta \geq 3 \times 10^4$, which in terms of the Reynolds number reads $Re \geq 1.4 \times 10^4$. For smaller values of $\beta$, the $k$-$\varepsilon$ model shows variations larger than 1%, which did not reduce after refining the mesh. In contrast, the LVEL model, was able to attain a variation smaller than 1% in the whole range of values $1 \times 10^4 \leq \beta \leq 1 \times 10^{12}$. The resulting range of Reynolds number was $1375 \leq Re \leq 4.6 \times 10^7$. The LVEL model was also able to reach smaller variations than the $k$-$\varepsilon$ model for the meshes considered here.

Table 6.1 shows the CPU times required for solving the $k$-$\varepsilon$ and LVEL models, for the 17 values of the volume force $\beta$ displayed in Figure 6.7, and for various mesh sizes. The $k$-$\varepsilon$ model did not converge for the coarse mesh of $16 \times 16$ elements. On the other hand, the LVEL model was able to find a solution for this mesh. From this table, we also can see that the LVEL model takes about half of the CPU time of the $k$-$\varepsilon$ model to find a solution.

For this particular case, we observe a speed up factor of about two. But this is not the whole story. In fact, it is important to point out that the zeroth order continuation with respect to the volume force term $\beta$ was crucial for the $k$-$\varepsilon$ model. In order to find a solution for a high value of $\beta$, the $k$-$\varepsilon$ model required to first solve the problem for a lower Reynolds number and then slowly increase the Reynolds number until the
Relative variation in the computed volumetric flow rate $Q$ as function of the applied volume force $\beta$ for various mesh sizes. Figure 6.7(a) shows the results for the $k$-$\varepsilon$ model and Figure 6.7(b) for the LVEL model. The variations in the mesh are indicated in the legend. The type of mesh used for both computations was a logarithmically spaced quadrilateral mesh as depicted in Figure 6.5.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>$16 \times 16$</th>
<th>$32 \times 32$</th>
<th>$48 \times 48$</th>
<th>$64 \times 64$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k$-$\varepsilon$ [s]</td>
<td>–</td>
<td>701</td>
<td>1697</td>
<td>3787</td>
</tr>
<tr>
<td>LVEL [s]</td>
<td>173</td>
<td>395</td>
<td>888</td>
<td>1801</td>
</tr>
</tbody>
</table>

Table 6.1: CPU times for solving the $k$-$\varepsilon$ and LVEL system of equations, for various meshes. The elements in the axial direction were distributed linearly, while the elements in the radial direction were distributed logarithmically (see Figure 6.5). The time was measured in seconds [s].

desired Reynolds number was reached, otherwise the Newton solver would fail. In contrast, the LVEL model was more robust and the method did converge without the help of the continuation method. We were able to find a solution for Reynolds numbers up to $Re = 1 \times 10^7$, without having to use the zeroth order continuation method. This is an advantage of the LVEL method from a practical point of view, for example, if one is interested in the solution at a high Reynolds number only. In such a situation, the $k$-$\varepsilon$ model, will require to solve the equations several times with the continuation method in order to find the solution, while the LVEL model must be solved only once.

Table 6.2 shows the CPU times needed to find a solution for a single given value of the volume force $\beta$ without the continuation method. With the $k$-$\varepsilon$ model it was possible to find a solution in such a way, only for limited values of $\beta$. For the values of $\beta$ showed in Table 6.2, the $k$-$\varepsilon$ model could not be solved directly and it required the continuation method. In contrast, it was possible to find a solution directly for the LVEL model. Solving the LVEL model for $\beta = 10^{10}$ takes more CPU time than solving for $\beta = 10^4$, how-
Table 6.2: CPU times required for solving the LVEL model without the continuation method, for two values of the volume force $\beta$. The results were computed for 4 different meshes.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>15 x 15</th>
<th>30 x 30</th>
<th>45 x 45</th>
<th>60 x 60</th>
<th>$\beta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time [s]</td>
<td>11</td>
<td>26</td>
<td>60</td>
<td>110</td>
<td>$10^4$</td>
</tr>
<tr>
<td>Time [s]</td>
<td>16</td>
<td>48</td>
<td>94</td>
<td>291</td>
<td>$10^{10}$</td>
</tr>
</tbody>
</table>

ever, some valuable CPU time was saved by avoiding using the continuation method and the CPU time did not increase very rapidly.

6.5.4 Experimental Validation for Straight Pipes

In the present section we provide validation of the $k$-$\varepsilon$ and the LVEL model by comparing the results to experimental values for fully developed turbulent flow in a smooth straight pipe. For the comparison, we use three sets of experimental data.

The first source corresponds to experimental data from the Princeton research group, for which they used compressed air in their Superpipe experiments. We consider the friction factor data by McKeon and Swanson et.al. [68], but we will refer to these data as the Princeton data set. The second source corresponds to experiments by the research group from Oregon. For their experiments, they used several room temperature gases. In particular we consider the experiments with liquid helium by Swanson and Julian et.al. [118], we will refer to these experiments as the Oregon data set. Joseph et.al. [51], compared the Princeton and Oregon data sets with effectively smooth pipe data, which they selected from the original experiments by Nikuradse [78]. A good agreement between the three data sets was found. The third data set corresponds to the experiments in the range $3.1 \times 10^4 \leq Re \leq 3.5 \times 10^7$ by Zagarola and Smits et.al [129].

Figure 6.8 shows a comparison between the experimental data sets mentioned above and the numerical results obtained with our turbulence models. Figures 6.8(a) and 6.8(b) show the comparison with the $k$-$\varepsilon$ and LVEL models, respectively. Both models provide a reasonably accurate approximation in the whole range of Reynolds numbers studied here. The $k$-$\varepsilon$ model slightly underestimates the friction factor in the range $3520 \leq Re \leq 5 \times 10^4$. The highest error is attained at $Re = 7192$, with a relative error of about 9%. The LVEL model slightly overestimates the friction factor in the same region. In this case the maximum relative error is about 10%. The maximum relative errors mentioned above do not consider the laminar and transition regions, i.e., we restricted the error calculation to the region $Re \geq 3500$.

Figures 6.8(c) and 6.8(d) show a zoom in of the previous plot in the high Reynolds number region $3.1 \times 10^4 \leq Re \leq 3.5 \times 10^9$. The $k$-$\varepsilon$ results are closer to the data set from
6.5 Numerical Analysis

Figure 6.8: Comparison of the k-ε and LVEL models with experimental values for fully developed turbulent flows in a straight pipe. Figure 6.8(a) shows the comparison with the k-ε model and Figure 6.8(b) shows the comparison for the LVEL model. The Oregon data set was taken from [118], the Princeton data from [68] and the Zagarola Smits from [129]. Figures 6.8(c) and 6.8(d) show a zoom in of the plots above in the range \(3.1 \times 10^4 \leq \text{Re} \leq 3.5 \times 10^6\).

Princeton from reference [68], while the LVEL model is closer to the data set by Zagarola et.al. [129]. We can see that both models work properly also in this high Reynolds number regime.

6.5.5 Comparison for a Corrugated Pipe

Finally we conclude with a comparison between the proposed turbulence models and the experiments for a corrugated pipe from Deiber et.al. [25]. The pipe used for the experiment was a sinusoidal pipe described as in Figure 3.14, with parameters \(a = 0.8571\), \(L = 8.9714\) and \(R_w(0) = 1\). Figure 6.9 shows a plot of the friction factor versus the Reynolds number. The red circles correspond to the experimental values by Deiber et.al. As we can see the data set shows transition towards turbulent flow for a Reynolds number...
of about $Re = 800$. Starting from a Reynolds number of about $Re = 1400$, the plot suggest that the flow is already turbulent, however precisely at this transition region, high variations between experiments are expected.

The blue crosses show the data points from the Oregon experiments for a straight pipe. This data set was included for reference. The magenta line shows the friction factor computed with the $k$-$\varepsilon$ model and the green lines shows the friction factor computed with the LVEL model. The computations done with the LVEL model have a relative error of about $20\%$ in the region $Re \geq 1400$ (i.e. in the turbulent region). The computations done with the $k$-$\varepsilon$ model have an error of about $50\%$ for this experiment. The results which we obtained with the LVEL model, give us an indication of the value of the friction factor for this particular example.

The large discrepancy between the experimental data and the numerical results of the $k$-$\varepsilon$ model could be related to the usage of wall-functions for imposing the boundary conditions at the physical wall. We should keep in mind that a pipe with corrugated walls is prompt to induce separated flows. Separated flows are well known as an “Achilles heel” for many turbulence models [83].

An alternative to try to alleviate this problem is to derive alternative laws of the wall which take into account wall-shape. The idea would be to improve the description of the flow for separated flows [1]. However, in order to answer this question precisely, further experiments an validations for turbulent flow in corrugated pipes need to be performed. It is important to keep in mind, that even the case of turbulent flow in a straight rough pipe is not fully understood and it is still a subject of current research [69,107].
Chapter 7

Conclusions and Recommendations

The study of this thesis was motivated by the practical need to describe flow and heat transfer in corrugated pipes. The walls of these type of pipes have corrugations which provide them with local stiffness and flexibility in the long scale. These characteristics make corrugated pipes appealing to use in applications such as offshore LNG transfer, cryogenic engineering, domestic appliances, etc. However, a complex wall-shape increases the difficulty of predicting the flow and heat transfer. The need for simplified models and efficient numerical methods for addressing this problem was the focus of this thesis. The present thesis combined both, numerical and analytical techniques in order to achieve this goal. The following achievements were obtained in this thesis.

1. We provided a framework for the governing equations of fluid flow and heat transfer. First we derived the Boussinesq equations and on the basis of these equations, we obtained the various limiting cases which we treated in this thesis, namely, isothermal laminar flow, non-isothermal laminar flow and turbulent flow. In the case of non-isothermal laminar flow we considered the cases of forced and natural convection.

2. We obtained integral formulas for the friction factor in axially symmetric periodic pipes, in the form of integrals. The first formula applies for periodic components and it was obtained by integrating the momentum equations and exploiting periodicity. For arbitrary components, we derived a generalization of the Bernoulli equation. This generalization allows us to represent the loss coefficients via another integral formula involving the entropy production in the flow. This representation can be used to compute the loss coefficients of in any other type of
components such as bends, diffusers, etc.

3. We developed a very efficient analytical formula for estimating the friction factor in axially symmetric periodic pipes. The formula was obtained by combining the integral representation of the friction factor with asymptotic solutions of the Navier-Stokes equations, which were derived by using the methods of slow variations. The analytical formula involves numerical integration in one dimension only, which makes it very efficient. When compared to a finite element model, we observed a speed up factor of about $10^3$. The accuracy of the formula was evaluated by comparison to experimental values and to numerical simulations, featuring pipes with sinusoidal corrugations. Our analytical formula was able to capture the dependency of the friction factor on the amplitude of the corrugations. This was not possible by considering the (0-th order) asymptotic solution alone. The analytical formula provided an error smaller than 10% for geometries with dimensionless amplitudes $a \leq 0.2$, provided that the period $L \geq 1.5$ and $Re \leq 750$. For larger amplitudes one requires either a small Reynolds number, or a large value of the period $L$.

4. In order to be able to compute the friction factor for geometries in which the analytical formula was not accurate, we introduced an efficient numerical model. The computational domain was reduced to just one period by applying periodicity decomposition to the pressure. The numerical model was used to systematically evaluate the accuracy of the analytical formula and also for addressing the problem of wall-shape design. Regarding wall-shape design, we performed a parametric study of the effects of wall-shape in the flow for geometries of cavity and slowly varying type. We showed that asymmetric wall-shapes (in the axial direction) have a smaller friction factor than the symmetric counterparts, for both types of geometries. In the case of slowly varying geometries, we observed an improvement in the flow rate of up to 120%.

5. In the case of non-isothermal laminar flow, we showed how to obtain an efficient numerical model for simulating laminar forced convection with constant wall heat flux. We showed how to use the periodicity decomposition technique for the pressure as well and for the temperature. The technique leads to a heat source term in the temperature equations which is obtained by an energy balance. Then we discussed the more challenging problem of natural convection. We considered an application involving the design of a thermosyphon loop for a cryogenic storage tank featuring corrugated pipes. We proposed a numerical model for solving this problem. Based on the proposed model, we showed that the maximum temperature inside the thermosyphon can be reduced by properly designing the wall-shape. In addition, the proposed model allowed us to see that a periodicity decomposition could not be applied. Qualitative information obtained from the simulations, was later exploited in order to develop a simplified model which offers a convenient alternative to the full numerical model.
6. We developed a simplified (homogenized) model for simulating natural convection in a vertical corrugated pipe. The simplified model was obtained by using formal homogenization. The starting point was to employ a multiple scale asymptotic expansion in the axial direction and a boundary layer type expansion in the radial direction. This was suggested by the solutions which we obtained from the full numerical model. The solution was split into an outer solution (valid away from the wall) and an inner solution (valid close to the wall). The outer and inner solution were then matched. The matching allowed us to obtain effective boundary conditions which we could impose on the homogenized domain. The effective boundary conditions are the key component in the homogenized model because they allow us to simplify the geometry of the domain while keeping the effects of wall-shape. The homogenized model correctly predicts the long scale behavior of the solutions and it is able to accurately predict averaged quantities such as the flow rate and the total heat transfer. The method works properly also for larger values of the wall heat-transfer coefficient. Over the whole range of heat transfer coefficients considered (0.01[W m\(^{-2}\) K\(^{-1}\)] \(\leq \alpha_w \leq 1\) [W m\(^{-2}\) K\(^{-1}\)]), the maximum relative error for the total heat transfer \(H\), Reynolds number \(Re\) and maximum Temperature \(T_{max}\) were 2\%, 3.7\% and 0.1\%, respectively. The homogenized model allows to simplify the geometry. This is very convenient from a computational point of view, for instance mesh generation becomes straightforward. Overall the homogenized model allows to accurately compute the solution in a fraction of the time of the full numerical model. The speed up factor strongly depends on the size of the whole domain and on the shape of the corrugations. We observed speed up factors from 5 up to 175.

7. In the case of isothermal turbulent flow, we presented and compared the two-equation \(k-\varepsilon\) turbulence model and the algebraic LVEL turbulence model. We discussed, the important role which wall-functions play in the description of wall-bounded turbulence. We presented a detailed description and implementation of the solution algorithm for the LVEL model. We addressed the accuracy of the models by comparing our numerical predictions with experiments from the literature for the case of fully developed turbulent flow in straight pipes. We also evaluated the numerical costs and other characteristics of both methods. The LVEL model shows to be more robust and efficient than the \(k-\varepsilon\) model. Both models are reasonably accurate for predicting the friction factor in straight pipe, the \(k-\varepsilon\) model show a maximum relative error of about 9\%, while the LVEL model showed a maximum relative error of about 10\%. The LVEL model was about 2 times faster than the \(k-\varepsilon\) model. An convenient characteristic of the LVEL model was that it was able to converge without the help of the continuation method with respect to the applied volume force. Comparison with experimental data for corrugated was provided.
In science there is always space for improvements and further developments and the topic of this thesis is no exception. A very natural idea would be to independently study (or incorporate) the effects of other type of phenomena, such as compressibility, acoustics, phase transition or boiling. But even without involving other type of phenomena, there is room for developments and we would like to point out couple of suggestions for future work.

The generalization of the Bernoulli formula presented in Chapter 3 can be used not only for periodic corrugated pipes, but for arbitrary conduits. It would be interesting to apply this formula and compare with experimental data for other type of components such as bends or diffusers. One could also combine this formula with the method of slow variations in order to come to analytical formulas which are also applicable to more general components.

For the case of forced convection treated in Chapter 4, it would be interesting to apply the analytical methods from Chapter 3, and see if it is possible to generalize the analytical formulas for the case of forced convection with constant wall heat flux. Also one could address the problem of forced convection with Dirichlet type boundary conditions.

In the case of natural convection, another interesting idea, would be to study the possibility of finding higher order approximations for the homogenized system of equations developed in Chapter 5.

The problem of turbulent flow is of course one of the biggest unresolved problems in science. This topic gives lots of opportunities to do research. Just to mention that the effects of roughness, even in the case of a straight pipe, are not fully understood yet [69,107]. In an analogous way as it was done in the laminar case, it would be interesting to consider the problem of designing a wall-shape for reducing the friction factor in turbulent flows.

Another idea would be to develop new wall-functions in order to improve the performance of the \( k-\varepsilon \) model near walls. One could also study the possibility of modifying the LVEL model by using a modified law of the wall. However, any development in this direction, should be accompanied by further comparison and validation with experimental data.
Appendix A

Tensor Calculations

A tensor (strictly speaking of order two) is a linear transformation of a Euclidean vector space into itself [48]. Given two vectors $a$ and $b$ in an Euclidean space, one can construct the tensor product $a \otimes b$. This is a mapping which assign a vector $x$ to the vector $a (b \cdot x)$, i.e.,

$$(a \otimes b) x := a (b \cdot x). \quad (A.1)$$

Given a basis $b_i$ for the Euclidean vector space, then the tensors $b_i \otimes b_j$ form a basis for the space of tensors. A tensor can be written as a matrix $\mathbf{A} = (a_{ij})$ with respect to this tensor basis, i.e.,

$$\mathbf{A} = \sum_{i,j} a_{ij} b_i \otimes b_j. \quad (A.2)$$

In our particular case we will concentrate in the 3-dimensional space described by the canonical Cartesian basis vectors $e_1, e_2$ and $e_3$, in which case a tensor $\mathbf{A}$ can be written as a $3 \times 3$ matrix $\mathbf{A} = (a_{ij})$. The identity tensor will be denoted by $\mathbf{I}$.

The divergence of a tensor $\mathbf{A}$ is a vector whose $i$-th component, in Cartesian coordinates $(x_1, x_2, x_3)$, is given by

$$(\nabla \cdot \mathbf{A})_i = \sum_j \partial_{x_j} a_{ij} \quad (i = 1, 2, 3). \quad (A.3)$$

The product of two tensors $\mathbf{A}$ and $\mathbf{B}$ produces a tensor $\mathbf{AB}$, whose components are
given by
\[(\mathcal{A}\mathcal{B})_{ij} := \sum_k a_{ik}b_{kj}.\] (A.4)

The double inner product of two tensors \(\mathcal{A}\) and \(\mathcal{B}\) produces a scalar \(\mathcal{A}: \mathcal{B}\), which can be evaluated as
\[\mathcal{A}: \mathcal{B} = \sum_{ij} a_{ij}b_{ij}.\] (A.5)

In turn, one can define the Frobenius norm of a second order tensor by
\[\|\mathcal{A}\| = (\mathcal{A}: \mathcal{A})^{1/2}.\] (A.6)

A.1 Viscous Stress Tensor for Incompressible Fluids

The incompressibility constraint \(\nabla \cdot \mathbf{u} = 0\) (obtained in the Boussinesq approximation (2.18)), allows us to simplify the Newton’s viscous stress tensor (2.6) as
\[
\mathbf{\tau} = 2\mu \mathbf{D} \text{ with } \mathbf{D} = \frac{1}{2} \nabla \mathbf{u} + \frac{1}{2} (\nabla \mathbf{u})^T, \tag{A.7}
\]
where by the term \(\nabla \mathbf{u}\), we actually mean \(\nabla \otimes \mathbf{u}\). The deformation velocity tensor can be written (using the canonical basis) as
\[
\mathbf{D}_{ij} = \frac{1}{2} \partial_{x_i} u_j + \frac{1}{2} \partial_{x_j} u_i. \tag{A.8}
\]

In matrix form, the viscous stress tensor takes the form
\[
\mathbf{\tau} = \begin{pmatrix}
\tau_{11} & \tau_{12} & \tau_{13} \\
\tau_{21} & \tau_{22} & \tau_{23} \\
\tau_{31} & \tau_{32} & \tau_{33}
\end{pmatrix} = \mu \begin{pmatrix}
2\partial_{x_1} u_1 & \partial_{x_2} u_1 + \partial_{x_1} u_2 & \partial_{x_3} u_1 + \partial_{x_1} u_3 \\
\partial_{x_1} u_2 + \partial_{x_2} u_1 & 2\partial_{x_2} u_2 & \partial_{x_3} u_2 + \partial_{x_2} u_3 \\
\partial_{x_1} u_3 + \partial_{x_3} u_1 & \partial_{x_2} u_3 + \partial_{x_3} u_2 & 2\partial_{x_3} u_3
\end{pmatrix}.\]
Using (A.8) and the definition of the divergence of a tensor we get

\[(\nabla \cdot \tau)_i = \mu (\nabla \cdot 2\mathbf{D})_i\]

\[= \mu \sum_j \partial_{x_j} \left( \partial_{x_j} u_i + \partial_{x_i} u_j \right)\]

\[= \mu \sum_j \partial_{x_j} x_i + \sum_j \partial_{x_i} x_j = \mu \nabla^2 u_i + \partial_{x_i} (\nabla \cdot \mathbf{u})\]

\[= \mu \nabla^2 u_i.\]

In the previous derivation we exchanged the order of differentiation in the fourth equality and we used that \(\nabla \cdot \mathbf{u} = 0\). Thus, for incompressible flows we have the identity (rewriting the previous equation in vector form)

\[\nabla \cdot \tau = \mu \nabla^2 \mathbf{u}. \quad (A.9)\]

In a similar manner, we can elaborate the expression for the viscous dissipation term \(\tau : \nabla \mathbf{u}\) in the entropy equation (2.72). To do this we decompose \(\nabla \mathbf{u} = \mathbf{D} + \Omega\), where \(\Omega := \frac{1}{2} \left( \nabla \mathbf{u} - (\nabla \mathbf{u})^T \right)\) and \(\mathbf{D}\) is defined as in (A.7). Using this decomposition we can compute

\[\tau : \nabla \mathbf{u} = 2\mu \mathbf{D} : (\mathbf{D} + \Omega) \quad (A.10)\]

\[= 2\mu \mathbf{D} : \mathbf{D} \quad (A.11)\]

\[= \frac{\mu}{2} \| \nabla \mathbf{u} + (\nabla \mathbf{u})^T \|^2, \quad (A.12)\]

where the second equality follows because the double inner product of a symmetric tensor (in this case \(\mathbf{D}\)) with an antisymmetric tensor (in this case \(\Omega\)) is always zero. To see this, we can take a symmetric tensor \(\mathbf{A} = (a_{ij})\) (i.e., \(a_{ij} = a_{ji}\)) and an antisymmetric one \(\mathbf{B} = (b_{ij})\) (i.e., \(b_{ij} = -b_{ji}\)), then

\[\mathbf{A} : \mathbf{B} = \sum_{i > j} a_{ij} b_{ij} + \sum_{i < j} a_{ij} b_{ij} \]

\[= \sum_{i > j} a_{ij} b_{ij} + \sum_{j < i} a_{ji} b_{ji} \]

\[= \sum_{i > j} a_{ij} b_{ij} - a_{ij} b_{ij} \]

\[= 0,\]

where in the second equality we exchanged the summation indices of the second term, and in the third equality we used the symmetry and antisymmetric properties of \(\mathbf{A}\) and \(\mathbf{B}\) respectively.
Appendix B

Cylindrical Coordinate System

In this appendix we include some useful identities involving the cylindrical coordinate system. We are mainly interested in writing various differential operators in cylindrical coordinates, such as $\nabla p$ or $\tau : \nabla u$. This can be done in a straightforward way once we have an expression for $\nabla$ in cylindrical coordinates, and once we know the spatial derivatives of the unit vectors in cylindrical coordinates. For further information we refer the reader to [8, 48]. In cylindrical coordinates a point is located by specifying the values of the variables $r$, $\phi$ and $x$. These cylindrical coordinates are related to the Cartesian coordinates by

$$
x_1 = r \cos \phi, \quad r = \sqrt{x_1^2 + x_2^2}, \quad (B.1a)
$$
$$
x_2 = r \sin \phi, \quad \tan \phi = x_2 / x_1, \quad (B.1b)
$$
$$
x_3 = x, \quad x = x_3, \quad (B.1c)
$$

with $r \geq 0$, $0 \leq \phi < 2\pi$ and $x \in \mathbb{R}$. By using the chain rule, we can convert derivatives with respect to the Cartesian variables $x_1$, $x_2$ and $x_3$, into derivatives with respect to $r$, $\phi$ and $x$, namely

$$
\partial_{x_1} = \cos \phi \partial_r - \frac{\sin \phi}{r} \partial_\phi, \quad (B.2a)
$$
$$
\partial_{x_2} = \sin \phi \partial_r + \frac{\cos \phi}{r} \partial_\phi, \quad (B.2b)
$$
$$
\partial_{x_3} = \partial_x. \quad (B.2c)
$$
Let \( \mathbf{e}_1, \mathbf{e}_2 \) and \( \mathbf{e}_3 \) be the canonical orthonormal basis vectors. Then the cylindrical basis vectors \( \mathbf{e}_r, \mathbf{e}_\phi \) and \( \mathbf{e}_x \) can be written as

\[
\begin{align*}
\mathbf{e}_r &= \cos \phi \mathbf{e}_1 + \sin \phi \mathbf{e}_2, \\
\mathbf{e}_\phi &= -\sin \phi \mathbf{e}_1 + \cos \phi \mathbf{e}_2, \\
\mathbf{e}_x &= \mathbf{e}_3.
\end{align*}
\]

We can also express the Cartesian unit vectors in terms of the cylindrical basis vector, this yields

\[
\begin{align*}
\mathbf{e}_1 &= \cos \phi \mathbf{e}_r - \sin \phi \mathbf{e}_\phi, \\
\mathbf{e}_2 &= \sin \phi \mathbf{e}_r + \cos \phi \mathbf{e}_\phi, \\
\mathbf{e}_3 &= \mathbf{e}_x.
\end{align*}
\]

Vectors and tensors can be decomposed into components with respect to the cylindrical coordinates, for example, the position vector \( \mathbf{x} \) and velocity vector \( \mathbf{u} \), have the following form

\[
\begin{align*}
\mathbf{x} &= r \mathbf{e}_r + x \mathbf{e}_x, \\
\mathbf{u} &= u_r \mathbf{e}_r + u_\phi \mathbf{e}_\phi + u_x \mathbf{e}_x,
\end{align*}
\]

where \( u_r, u_\phi \) and \( u_x \) are called, the radial, azimuthal and axial velocity components, respectively.

From (B.3), we can obtain expressions for the spatial derivatives of the unitary cylindrical vectors \( \mathbf{e}_r, \mathbf{e}_\phi \) and \( \mathbf{e}_x \), and these read

\[
\begin{align*}
\partial_r \mathbf{e}_r &= 0, \\
\partial_r \mathbf{e}_\phi &= 0, \\
\partial_r \mathbf{e}_x &= 0, \\
\partial_\phi \mathbf{e}_r &= \mathbf{e}_\phi, \\
\partial_\phi \mathbf{e}_\phi &= -\mathbf{e}_r, \\
\partial_\phi \mathbf{e}_x &= 0, \\
\partial_x \mathbf{e}_r &= 0, \\
\partial_x \mathbf{e}_\phi &= 0, \\
\partial_x \mathbf{e}_x &= 0.
\end{align*}
\]

\section*{B.1 Differential Operators}

We can use the previous relations in order to rewrite the gradient operator \( \nabla \) in cylindrical coordinates. We start by recalling the gradient operator in Cartesian coordinates, namely \( \nabla = \mathbf{e}_1 \partial_{x_1} + \mathbf{e}_2 \partial_{x_2} + \mathbf{e}_3 \partial_{x_3} \). By substituting (B.4), (B.2) and simplifying the expression, we obtain the gradient in cylindrical coordinates

\[
\nabla = \mathbf{e}_r \partial_r + \mathbf{e}_\phi \frac{1}{r} \partial_\phi + \mathbf{e}_x \partial_x.
\]
This expression can be used for obtaining all required differential operators in cylindrical coordinates. Let $p$ and $u = u_r e_r + u_\phi e_\phi + u_x e_x$ be smooth functions of the cylindrical coordinates $r$, $\phi$ and $x$. Then the gradient $\nabla p$, Laplacian $\nabla^2 p$ and divergence $\nabla \cdot u$, are given by

\[
\nabla p = \partial_r p e_r + \frac{1}{r} \partial_\phi p e_\phi + \partial_x p e_x, \quad (B.8a)
\]

\[
\nabla^2 p = \frac{1}{r} \partial_r (r \partial_r p) + \frac{1}{r^2} \partial_\phi^2 p + \partial_{xx} p, \quad (B.8b)
\]

\[
\nabla \cdot u = \frac{1}{r} \partial_r (r u_r) + \frac{1}{r} \partial_\phi u_\phi + \partial_x u_x. \quad (B.8c)
\]

Formula (B.7) can be used for operating on vectors as well, provided that we use (B.6) in order to differentiate any unit vectors on which $\nabla$ operates and keeping in mind that when we write $\nabla \mathbf{u}$, we actually mean $\nabla \otimes \mathbf{u}$. For example, $\nabla u$ in cylindrical coordinates can be computed in the following way

\[
\nabla u = e_r \otimes \partial_r (e_r u_r + e_\phi u_\phi + e_x u_x) + e_\phi \otimes \frac{1}{r} \partial_\phi (e_r u_r + e_\phi u_\phi + e_x u_x)
\]

\[
+ e_x \otimes \partial_x (e_r u_r + e_\phi u_\phi + e_x u_x)
\]

\[
= \partial_r u_r e_r \otimes e_r + \partial_r u_\phi e_\phi \otimes e_\phi + \partial_r u_x e_x \otimes e_x + e_\phi \otimes \frac{1}{r} \left( e_r \partial_\phi u_r + u_r e_r \right)
\]

\[
+ e_\phi \otimes \frac{1}{r} \left( e_\phi \partial_\phi u_\phi - u_\phi e_\phi \right) + \frac{1}{r} \partial_\phi u_\phi e_\phi \otimes e_x + \partial_x u_r e_r \otimes e_r + \partial_x u_\phi e_\phi \otimes e_\phi + \partial_x u_x e_x \otimes e_x,
\]

which in matrix form reads

\[
\nabla u = \begin{pmatrix}
\partial_r u_r & \partial_\phi u_r & \partial_x u_r \\
\frac{1}{r} \partial_\phi u_r - \frac{u_\phi}{r} & \frac{u_\phi}{r} + \frac{1}{r} \partial_r u_\phi & \frac{1}{r} \partial_\phi u_x \\
\partial_x u_\phi & \partial_\phi u_\phi & \partial_x u_x
\end{pmatrix}.
\quad (B.9)
\]

### B.2 Viscous Stress Tensor

By using (A.7) and (B.9), we can write the stress tensor $\mathbf{\tau}$ in cylindrical coordinates. In matrix form, this reads

\[
\mathbf{\tau} = \begin{pmatrix}
\tau_{rr} & \tau_{r\phi} & \tau_{rx} \\
\tau_{r\phi} & \tau_{\phi\phi} & \tau_{\phi x} \\
\tau_{rx} & \tau_{\phi x} & \tau_{xx}
\end{pmatrix} = \mu \begin{pmatrix}
2 \partial_r u_r & r \partial_r \left( \frac{u_\phi}{r} \right) + \frac{1}{r} \partial_\phi u_r & \partial_x u_r + \partial_r u_x \\
\partial_\phi u_r & \frac{1}{r} \partial_\phi u_r + \frac{1}{r} \partial_\phi u_r & 2 \left( \frac{1}{r^2} \partial_\phi u_\phi + \frac{u_\phi}{r} \right) \\
\partial_x u_\phi + \partial_r u_x & \partial_\phi u_\phi + \frac{1}{r} \partial_\phi u_x & 2 \partial_x u_x
\end{pmatrix}.
\quad (B.10)
\]
In particular, we are interested in an expression of the viscous dissipation term \( \tau : \nabla u \) in cylindrical coordinates. By using (B.10) and (B.9) we find that

\[
\tau : \nabla u = \mu \left\{ 2 \left( \frac{1}{r} \partial_r u_r \right)^2 + \left( \frac{1}{r} \partial_\phi u_\phi + \frac{u_r}{r} \right)^2 + (\partial_x u_x)^2 \right\} \\
+ \left[ \left( \frac{1}{r} \partial_r \left( \frac{u_\phi}{r} \right) + \frac{1}{r} \partial_\phi u_r \right)^2 + (\partial_x u_r + \partial_r u_x)^2 + \left( \frac{1}{r} \partial_\phi u_\phi + \frac{1}{r} \partial_\phi u_x \right)^2 \right] \right\}. \tag{B.11}
\]

In the particular case of axially symmetric flows (i.e., \( u_\phi = 0 \), and \( \partial_\phi = 0 \)), the previous expression simplifies to

\[
\tau : \nabla u = \mu \left\{ 2 \left( \frac{1}{r} \partial_r u_r \right)^2 + \left( \frac{u_r}{r} \right)^2 + (\partial_x u_x)^2 \right\} + (\partial_x u_r + \partial_r u_x)^2 \right\}. \tag{B.12}
\]

### B.3 Boussinesq Equations

By using the previous calculation rules, one can rewrite the Boussinesq system of equations (2.29) in cylindrical coordinates; they read

continuity:
\[
\frac{1}{r} \partial_r (ru_r) + \frac{1}{r} \partial_\phi u_\phi + \partial_x u_x = 0, \tag{B.13a}
\]

\( r \)-momentum:
\[
\rho \left( \partial_t u_r + u_r \partial_r u_r + \frac{u_\phi}{r} \partial_\phi u_r - \frac{u_r}{r} + u_x \partial_x u_r \right) = -\partial_r p \\
+ \mu \left[ \frac{1}{r} \partial_r (r \partial_r u_r) - \frac{u_r}{r^2} + \frac{1}{r} \partial_\phi u_r - \frac{2}{r^2} \partial_r u_\phi + \partial_x u_r \right] - \rho \alpha_c (T - T_{ref}) g \cdot e_r, \tag{B.13b}
\]

\( \phi \)-momentum:
\[
\rho \left( \partial_t u_\phi + u_r \partial_r u_\phi + \frac{u_\phi}{r} \partial_\phi u_\phi + \frac{u_r}{r} + u_x \partial_x u_\phi \right) = \frac{1}{r} \partial_\phi p \\
+ \mu \left[ \frac{1}{r} \partial_r (r \partial_r u_\phi) - \frac{u_\phi}{r^2} + \frac{1}{r^2} \partial_\phi u_r + \frac{2}{r^2} \partial_r u_\phi + \partial_x u_\phi \right] - \rho \alpha_c (T - T_{ref}) g \cdot e_\phi, \tag{B.13c}
\]

\( x \)-momentum:
\[
\rho \left( \partial_t u_x + u_r \partial_r u_x + \frac{u_\phi}{r} \partial_\phi u_x + u_x \partial_x u_x \right) = -\partial_x p \\
+ \mu \left[ \frac{1}{r} \partial_r (r \partial_r u_x) + \frac{1}{r^2} \partial_\phi u_x + \partial_x u_x \right] - \rho \alpha_c (T - T_{ref}) g \cdot e_x, \tag{B.13d}
\]

temperature:
\[
\rho C_p \left( \partial_t T + u_r \partial_r T + \frac{u_\phi}{r} \partial_\phi T + u_x \partial_x T \right) \\
= \kappa \left[ \frac{1}{r} \partial_r (r \partial_r T) + \frac{1}{r^2} \partial_\phi T + \partial_x T \right] + \tau : \nabla u, \tag{B.13e}
\]

where \( \tau : \nabla u \) (usually neglected in (B.13e), see Section 2.4), is given by (B.11).
Bibliography


Index

asymptotic ansatz, 104
asymptotic expansion, 53, 104
average axial velocity, 26, 47, 131
average heat flux, 87, 131
Bernoulli’s equation, 32
boundary condition, 47, 86
axial symmetry, 100
constant wall heat flux, 75, 76
Dirichlet, 24, 86
effective, 98, 112, 114, 119, 120
natural, 25
Neumann, 25, 86
no-slip, 25, 76, 87, 100
periodic, 25
Robin, 100
turbulent flow, 165
boundary layer
ansatz, 103
thermal, 89
velocity, 102
Boussinesq
approximation, 15, 20, 79, 82
continuity equation, 17
dimensionless system of equations, 99
energy equation, 19
equations in cylindrical coordinates, 184
momentum equations, 18
Boussinesq assumption, 149
Brinkman number, 21, 23
buoyancy speed, 23
Cauchy equations, 12
conductivity, 14
conservation equations, 12
contraction region, 63
convection
forced, 22, 23, 74, 75
natural, 23, 24, 74, 85
correction factor, 57
deformation velocity tensor, 14
dissipation of mechanical energy, 32, 35, 43
divergence
in cylindrical coordinates, 104
in inner region, 105
in outer region, 104
Eckert number, 21
entropy equation, 37
entropy production, 37, 38
expansion region, 63
Fourier’s heat flux law, 14
friction factor, 26, 27, 29, 31, 32, 40, 43, 44, 47, 49, 51, 57, 59, 170, 171
friction velocity, 154
fully developed flow, 45
geometry, 64
axially symmetric, 28
cavity configuration, 41, 49, 63, 64
slowly varying configuration, 41, 68
slowly varying geometry, 63
gradient
in cylindrical coordinates, 104, 182
in inner region, 105
in outer region, 104
Grashof number, 21
  modified, 87, 131
heat flux, 87, 131
homogenization
  cell problem, 109
  corrugated boundary, 101
  full numerical solution, 125
  homogenized solution, 125
  inner solution, 103
  method, 97
  of boundary conditions, 107, 120
  outer solution, 103, 105, 115
  reduced numerical solution, 125
  speed up, 140, 141
homogenized
  temperature equation, 112
  boundary, 101
  Boussinesq system of equations, 120
  domain, 101, 122
  flow equation, 117
hydrodynamic entrance region, 45
hydrostatic pressure, 18
incompressibility, 14
kinematic viscosity, 18
laminar flow, 26
law of the wall, 152, 155
  logarithmic, 152, 155
  Spalding’s, 158
  velocity profile, 152
  wall units, 154
LNG, 3, 80, 82, 84
local Laplace equation, 112
local radius, 104
loss coefficient, 32, 43
mechanical energy equation, 33
mesh
  boundary layer, 167
  independence, 49
  mapped, 49
mesh generation, 121
method of matched asymptotic expansions, 103
  boundary layer variable, 103
  matching for derivatives, 111
  matching principle, 109
  Prandtl matching principle, 110
method of multiple scales, 103
  fast variable, 103
method of slow variations, 51, 52
minor losses, 31
Moody diagram, 28
multiscale behavior, 102
Navier-Stokes equations, 15, 29, 41, 52
Newtonian fluid, 14
non-isothermal flow, 73
Nusselt number, 101
Péclet number, 21
pipe
  corrugated, 2, 4, 39, 42, 45, 75, 77, 79,
    85, 89, 97–99, 163, 171
  periodic, 53
  rough, 172
  sinusoidal, 48, 60, 63, 121, 171
  straight, 26, 31, 32, 43–45, 64, 77, 89,
    91, 166, 167, 170
Poiseuille flow, 27
Prandtl number, 21
pressure drop, 26, 27, 30
  due to skin friction, 58
  due to wall pressure forces, 31, 58
pressure drop
  due to skin friction, v
  pressure gradient, 44
  average, 46
quasi-incompressibility, 17, 19
Rayleigh number, 21
relative error, 60, 139, 168, 172
Reynolds averaged Navier-Stokes equations, 143, 144
momentum equations, 147
  time-dependent, 148
Reynolds decomposition, 144
Reynolds number, 21, 47, 91, 131
  local estimation, 161
Reynolds number local, 160
Reynolds stress tensor, 147
slowly varying
  leading order solution, 55
  variables, 53
stress tensor
  Newton’s viscous, 14
  viscous, 13

temperature
  maximum, 131
temperature equation, 78
temperature gradient
  average, 78, 79
tensor, 177
  divergence of, 177
  double inner product, 178
  Frobenius norm, 178
  product, 177
thermosyphon, 85
thermosyphon loop, 79, 80
time average, 144
time averaging, 145
turbulence $k$-$\varepsilon$ model, 143
  boundary conditions, 155, 157
  dissipation of turbulent kinetic energy, 150
  transport equations, 151
  turbulent kinetic energy, 147
turbulence LVEL model, 143, 158, 165
  solution algorithm, 162
turbulent flow, 45, 143
turbulent pressure, 149
turbulent viscosity, 159

velocity profile
  parabolic, 44
viscosity, 14

viscous dissipation, 21, 38, 179
  in cylindrical coordinates, 184
volumetric flow rate, 47, 131
volumetric thermal expansion coefficient, 16
wall shear stress, 153
wall-shape, 63
design, 70, 94
The transport of fluids through pipes is a very common practical need. Water is transported through complex networks of pipelines for domestic use and oil is transported for hundreds or even thousands of kilometers along pipelines. Despite their antiquity, there are still research and developments on the design of pipes and piping related technologies. Some properties, such as portability and flexibility, are convenient characteristics of the so-called corrugated pipes. These are pipes with non-straight walls, i.e., with corrugations. The corrugations provide local stiffness and flexibility on the long scale to the pipe. These characteristics make these type of pipes convenient in several application areas such as offshore LNG (Liquefied Natural Gas) transfer, cryogenic engineering, domestic appliances, etc. However, the introduction of the corrugations increases the difficulty of simulating flow and heat transfer.

The present thesis addresses the development of efficient models and numerical methods for simulating fluid flow and heat transfer in arbitrarily shaped pipes. The present work combines both, numerical and analytical techniques. We start by introducing the governing equations of fluid flow and heat transfer. We present the Boussinesq approximation and based on it we derive the governing equations for isothermal flow and non-isothermal flow. In the case of non-isothermal flow, we consider the limiting cases of forced and natural convection. We put special attention to the computation of the losses of mechanical energy and derive integral expressions for the friction factor (for periodic corrugated pipes) and for the loss coefficient (for arbitrary conduits).

In the case of isothermal laminar flow we develop a very efficient analytical formula for computing the friction factor in slowly varying pipes. For more general geometries, we present an efficient numerical model which uses a periodicity decomposition in order to reduce the numerical domain to just one period. We use the numerical model for systematically evaluating the accuracy of the analytical formula. Based on the presented models, we also address the problem of wall-shape design.

We start the discussion of non-isothermal flow with the problem of forced convection. In particular, we consider laminar forced flow with constant prescribed heat flux at the
walls. As in the case of isothermal flow, we use periodicity decomposition for reducing the computational domain. Then we proceed with the discussion of natural convection. For this case, we first take a practically oriented approach and concentrate on an industrial application involving a cryogenic storage tank featuring a thermosyphon loop. We present a numerical model to simulate the involved phenomena. With this numerical model, we show that it is possible to optimize the wall-shape of the thermosyphon. However the computational cost of such a numerical model are considerable. This happens mainly because the problem does not allow for a periodicity decomposition and the whole extension of the domain needs to be considered. In addition, the corrugations introduce multiple scales which further increase the computational requirements for handling this problem.

We provide a more efficient alternative for simulating natural convection by using the method of homogenization. The homogenization method allows us to replace the boundary conditions on a complex boundary by certain effective boundary condition on a homogenized (much simpler) boundary. This is advantageous from a computational point of view because the generation of an adequate mesh becomes straightforward and it is also easier to numerically solve the equations. At the same time, the effects of wall-shape are kept via the effective boundary conditions. The homogenized model is able to handle developing flows and can capture boundary layers. The homogenized model accurately predicts local and averaged quantities in a fraction of the costs of the direct numerical approach.

We continue with the case of isothermal turbulent flow. We first present the (RANS) Reynolds averaged Navier-Stokes equations. On this framework, we introduce two turbulence models: the two-equation $k$-$\varepsilon$ model and the algebraic LVEL model. We validate both models with experimental data and provide a comparison between both models.
Samenvatting

In de praktijk is er een grote behoefte aan vloeistoftransport door pijpleidingen. Zo wordt bijvoorbeeld water voor huishoudelijk gebruik vervoerd door complexe stelsels aan pijpleidingen, en wordt olie vervoerd door honderden, soms duizenden kilometers aan leidingen. Ondanks dat het idee al heel oud is wordt er nog altijd veel onderzoek gedaan naar het ontwerp van pijpleidingen en gerelateerde technologieën. De zogenaamde geribbelde pijpen, pijpen met een geribbelde wand, hebben belangrijke eigenschappen als draagbaarheid en flexibiliteit. Door de ribbels krijgen de pijpen op de lange schaal lokale stijfheid en flexibiliteit. Hierdoor zijn dit soort pijpen erg geschikt voor bepaalde toepassingen; onder andere voor het afdaling vervoer van vloeibaar aardgas, voor cryogene techniek, of voor huishoudelijke toepassingen. Desalniettemin is het voor geribbelde pijpen erg moeilijk om de stroming en warmteoverdracht door de pijpen te simuleren.

Dit proefschrift behandelt de ontwikkeling van efficiënte modellen en numerieke methoden om stroming en warmte-overdracht in pijpen van willekeurige vorm te simulieren. Met dit doel voor ogen worden numerieke methoden gecombineerd met analytische. Om te beginnen beschrijven we een algemeen raamwerk van beschrijvende vergelijkingen voor vloeistof en warmte-overdracht. We presenteren de Boussinesq-benadering, op basis waarvan we de beschrijvende vergelijkingen voor isotherme en niet-isotherme stroming afleiden. Voor de niet-isotherme stroming beschouwen we twee limietgevallen: gedwongen en vrije convectie. We besteden extra aandacht aan het berekenen van de hoeveelheid verloren mechanische energie, en leiden integraaluitdrukkingen af voor de wrijvingsfactor (voor periodiek geribbelde pijpen) en de verliescoëfficiënt (voor willekeurige pijpleidingen).

Voor isotherme en laminaire stroming ontwikkelen we een zeer efficiënte analytische formule waarmee de wrijvingsfactor in langzaam variërende pijpen kan worden berekend. Voor het geval van pijpen met een niet-langzaam variërende structuur presenteren we een efficiënt numeriek model, dat gebruik maakt van periodiciteitsdecompositie zodat het numerieke domein beperkt kan worden tot één periode. We gebruiken dit model om de nauwkeurigheid van de analytische methode te evalueren. Op basis van
deze modellen behandelen we ook hoe de wand van pijpleidingen ontworpen dienen te worden.

Voor niet-isotherme en laminaire stroming bekijken we twee limietgevallen: gedwongen en vrije convectie. Voor het geval van de gedwongen convectie bekijken we een vaste constante warmteflux langs de wand van de pijp. Net als in het geval van isotherme stroming gebruiken we periodiciteitsdecompositie om het numerieke domein te beperken. Voor het geval van vrije convectie bekijken we eerst een probleem uit de praktijk, betreffende een cryogene opslagtank met een thermosifonlus.

We beschrijven een numeriek model om dit praktijkprobleem te simuleren. We laten met dit model zien dat het mogelijk is om de vorm van de sifonwand te optimaliseren. De computationele kosten van dergelijke simulaties zijn echter aanzienlijk. Dit heeft vooral te maken met het feit dat er in dit geval geen periodiciteitsdecompositie kan worden gedaan, zodat de volledige uitbreiding van het domein moet worden meegegeven in de berekening. Bovendien zorgen de ribbels ervoor dat het probleem zich op meerdere schalen afspeelt, waardoor er nog meer rekenkracht vereist is om dit probleem op te lossen.

Met behulp van de homogenisatiemethode leveren we een efficiënter alternatief voor het simuleren van vrije convectieproblemen. Met deze methode kunnen we de randvoorwaarden op een ingewikkelde rand vervangen door effectieve randvoorwaarden op een eenvoudigere, gehomogeniseerde rand. Dit is computationeel gezien voordelig: het is eenvoudig om een adequaat rooster te construeren, en de vergelijkingen zijn makkelijker numeriek op te lossen. Het gehomogeniseerde model is in staat om ontwikkelende stromingen en randlagen te beschrijven, en het kan lokale en gemiddelde grootheden nauwkeurig voorspellen in slechts een fractie van de benodigde tijd vergeleken met de directe methode.

Acknowledgments

During these four years as a PhD student I made many friends and met lots of colleagues who made my life as a PhD an enjoyable experience. I would like to express my gratitude towards all of you for the important role you played for me to finish this thesis.

First and foremost, I would like to thank my promotors prof. Bob Mattheij and prof. Jacques Dam for giving me the opportunity to take part in such an interesting and challenging project. My promotors gave me freedom for organizing my work and at the same time they helped me to keep on track in the process of writing this thesis. I also would like to express my gratitude towards my copromotor dr. Jan ten Thije Boonkkamp for his support, friendliness and for carefully reviewing papers and this thesis.

I am grateful towards the members of my defense committee prof. Anton van Steenhoven, prof. Kees Vuik, prof. Jaap van der Vegt and towards the members of the extended committee prof. Barry Koren, prof. Harald van Brummelen and prof. Emile Aarts for their willingness and time devoted to read and evaluate this thesis.

My experience as a PhD would have not been such a memorable experience without the friendly and stimulating environment at the CASA group. I would like to thank Bas van der Linden for all the friendly discussions and for his help on setting many of the numerical simulations presented in this thesis. I am grateful to Kundan Kumar for the time we spent analyzing many of the derivations presented in Chapter 5 and for the pleasant non work related discussions. I would like to express my gratitude to Michiel Renger for his help translating the summary of this thesis into Dutch. I also thank dr. Sorin Pop, dr. Adrian Muntean, dr. Sjoerd Rienstra and prof. Mark Peletier for the fruitful conversations. I also would like to thank prof. Wil Schilders, dr. Jan ter Maten, dr. Michiel Hostenbach and dr. Joost Rommes for the opportunities to participate in conferences and courses in other scientific areas. Furthermore I thank Enna van Dijk and Marèse Wolfs van de Hurk for their kind treatment, help with administrative matters and for the organization of many refreshing social activities.
I thank my office mates Maria Rudnaya, Oleg Matveichuk and Martien Oppeneer for always keeping a nice atmosphere. I enjoyed being part of the organization of the CASA cooking events together with Maria and it was great to practice my Dutch with Martien. The conversations and jokes with Oleg were a pleasure and he was always willing to help me with all kind of matters. I would like to thank Erwin Vondenhoff and Peter int Panhuis for the role they played in the integration of all the PhD students and postdocs at the time when I joined the group. I am grateful to Lena Filatova and Oleg Matveichuk for being with me as paranymphs during the defense.

I also thank all my other colleagues and friends at CASA. Thank you Ali Etaati, Andriy Hlod, Agnieszka Lutowska, Akshay Iyer, Arpan Ghosh, Berkan Sasen, Badr Kaoui, Corien Prins, Carlo Mercuri, Darcy Hou, David Bourne, Davit Harutyunyan, Evgeniya Balmashnova, Eric Creusen, Godwin Kakuba, Giovanni Bonaschi, Hans Groot, Iason Zisis, Jos Maubach, Joep Evers, Lucia Scardia, Laura Astola, Mirela Darau, Mark van Kraaij, Maxim Pisarenco, Miguel Patricio Dias, Manh Hong Duong, Nicodemus Banagaaya, Neda Sepasian, Patrick van Meurs, Prasad Perlekar, Roxana Ionutiu, Ronald Rook, Rostyslav Polyuga, Sinatra Canggih, Sudhir Srivastava, Shruti Gumaste, Steffen Arnrich, Tasnim Fatima, Volha Shchetnikava, Valeriu Savcenco, Yabin Fan, Yves van Gennip, Yixin Lu, Zoran Ilievski, Zizheng Cao and everyone else who made my time in the group so enjoyable.

In addition I would like to acknowledge my friends and colleagues outside CASA. I particularly thank Edgar Resendiz for the long distance discussions and Israel Garza for designing the cover of this thesis. I also thank Antonino Simone, Edgar Resendiz, Gaetan Bison, Ivan Serna, Jan Willem Knopper, Jiri Cervenka, Mayla Bruso, Satyajit Wat Tamwar, Shona Yu and all my other friends with whom I have shared many experiences in life.

Last, but certainly not least, I would like to thank my family. My very supportive and kind parents Patricio and Maria Imelda who always showed me the best and most important things in life. My cheerful siblings Karol, Omar and Abril with whom life has always been a memorable fun experience. I would specially like to thank my recently formed family. My lovely and supportive wife Maria and my dear and lively son Patricio Piotr, you have been my source of inspiration and with you I am learning the meaning of being a husband and father. It is to my dear family that I dedicate this thesis.

Patricio Rosen

Curriculum Vitae

Patricio Ivan Rosen Esquivel was born on 7th July 1981 in Mexico City, Mexico. In 2000 he started his studies in the National University of Mexico (UNAM), where he completed two bachelor programs, one in Actuarial Science and a second one in Mathematics. In 2006 Patricio was awarded an Erasmus Mundus scholarship from the European Commission which allowed him to follow a double degree program in Industrial Mathematics in the Eindhoven University of Technology in the Netherlands and in the University of Kaiserslautern in Germany. He graduated cum laude from this program in the year 2008. He wrote his master’s thesis entitled “Factorization of Indefinite Systems Associated with RLC Circuits” under the supervision of prof.dr. W.H.A. Schilders and dr. J. Rommes.

From 2008 to 2012 he worked as a PhD researcher at the Eindhoven University of Technology in the Centre for Analysis Scientific Computing and Applications (CASA), under the supervision of prof.dr. R.M.M. Mattheij, prof.dr. J.A.M. Dam and dr. J.H.M. ten Thije Boonkkamp. The results of this research are presented in this dissertation.

In November 2012 he joined Shell Global Solutions as a Flow Assurance Engineer in the Multiphase Flow team of prof. Ruud Henkes within the PFAS group of dr. Loek Vreenegoor.