Friction factor estimation for turbulent flows in corrugated pipes with rough walls

Pisarenco, M.

Award date:
2007
Friction Factor Estimation
for Turbulent Flows
in Corrugated Pipes
with Rough Walls

by Maxim Pisarenco

Supervisors:
Prof. Dr. Robert Mattheij (TUE)
Dr. Ir. Arris Tijsseling (TUE)
Dr. Ir. Bas van der Linden (TUE)
Prof. Dr. Axel Klar (TU Kaiserslautern)

Eindhoven, August 2007
Acknowledgements

I would like to acknowledge Prof. Dr. Robert Mattheij, head of the Scientific Computing group, for offering me the possibility to work on this subject. I would also like to thank Dr. Ir. Arris Tijsseling for supervising this thesis. His advices, comments and support throughout this period were fundamental. Special thanks to Dr. Ir. Bas van der Linden, whose technical and scientific support has been extremely helpful.

I would like to express my esteem for Prof. Dr. Axel Klar from Technische Universität Kaiserslautern, who kindly agreed to give his valuable comments on my thesis and whose lectures were a delight to visit in my first year in Kaiserslautern.

I wish to express my gratitude to my friends and colleagues, Alexandru Mereacre, Serghei Zadorojnai, Andrian Ciobanu, Gheorghe Moisei, Lengxi Liu, Rajesh Kumar, Andis Dembovskis, Ivan Serna, Philip Reuterswärd, Clemens Zarzer, Erwin Karer, Tefa Kaisara, for sharing with me not only their knowledge but also many fun times.

I would also like to acknowledge the European Committee for its financial support of my Master studies in Kaiserslautern and Eindhoven through the Erasmus Mundus Scholarship program. These have been two wonderful years.

My eternal gratitude goes to my family for their prayers, their love, their patience and their support.
Summary

Non-metallic flexible pipes are widely used in industry. They are comprised of fabric wrapped over a spiral metal framework. Due to this construction, they respond very well to bending and they are cheaper and much easier to install than metal pipes. Because of the specific construction, the pipe walls are corrugated and the fabric which covers the steel spiral is much rougher than the wall of a metal pipe. In this investigation we are interested in estimating the friction factor for the flow in this type of pipes.

Two-equation turbulence models ($k - \epsilon$ and $k - \omega$) are used in the computations. The process of deriving these models is presented first. Then we look at turbulent boundary layers and the law of the wall which gives the velocity profile near the wall. Its use as a boundary condition is explained.

After the theoretical ideas have been exposed, we look at fully developed turbulent flow in a conventional pipe. Simulations are performed to validate the chosen models, boundary conditions and computational grids. Then a new boundary condition is implemented based on the "combined" law of the wall. It enables us to model the effects of roughness. The new boundary condition is validated by performing simulations of turbulent flow in rough pipes and comparing the computed friction factor to the one given by the Moody diagram.

Finally, turbulent flow in periodically corrugated (flexible) pipes is considered. New flow phenomena (such as flow separation) caused by the corrugation are pointed out and the essence of periodically fully developed flow is explained. The friction factor for different values of relative roughness of the fabric is estimated by performing a series of simulations. Conclusions are drawn based on the results of the computations. Some of them are that the $k - \epsilon$ model performs slightly better than the $k - \omega$ model in predicting both regular and separated flow and that the friction factor in a flexible corrugated pipe is mostly determined by the shape and size of the steel spiral, and not by the type of the fabric which is wrapped around the spiral.
Contents

1 Introduction 3
  1.1 Goals of the Investigation .......................... 3
  1.2 Descriptions of Turbulence .......................... 3
  1.3 Simulation of Turbulent Flows: DNS, LES and RANS .... 6
  1.4 Report Outline .................................. 9

2 Two-Equation Turbulence Models 11
  2.1 Equations Describing the Dynamics of Flow .......... 11
  2.2 Reynolds Averaging ................................ 12
  2.3 Eddy Viscosity Approximation ........................ 14
  2.4 The Reynolds Stress Equation ........................ 15
  2.5 The Turbulence Energy Equation ...................... 18
  2.6 The Turbulence Dissipation Equation and the $k - \epsilon$ Model 20
  2.7 The $\omega$-Equation and the $k - \omega$ Model .......... 21

3 Turbulent Boundary Layers 23
  3.1 The Turbulent Shear Layer Equations .................. 23
    3.1.1 Order of Magnitude Estimates ................... 24
    3.1.2 The Streamwise Momentum Equation ............... 25
    3.1.3 The Transverse Momentum Equation ............... 27
  3.2 Wall-Bounded Turbulent Flow ........................ 27
    3.2.1 The "Outer" Turbulent Boundary Layer .......... 28
    3.2.2 The "Inner" Turbulent Boundary Layer .......... 29
  3.3 Sublayers of the Constant Stress Layer ............... 32
  3.4 The Law of the Wall ................................ 35

4 The Law of the Wall as Boundary Condition 39
  4.1 Computational Approaches for the Near-Wall Region .... 39
  4.2 Boundary Conditions ................................ 39
  4.3 Thickness of the Wall Region ........................ 43

5 Turbulent Flow in Conventional Pipes 45
  5.1 Pipe Flow ...................................... 45
  5.2 Validation Tests for Pipe Flow and the Moody Diagram .... 47
  5.3 Simulation Setup .................................. 49
    5.3.1 Computational Domain Geometry .................. 49
    5.3.2 Boundary Conditions ............................ 50
    5.3.3 Meshing and Solution Procedure .................. 52
  5.4 Smooth Wall Validation ................................ 53
CONTENTS

5.5 Rough Wall Modeling .................................................. 55
5.6 Rough Wall Validation ............................................... 58

6 Friction Factor Computations for Corrugated Pipes .............. 61
6.1 Characteristics of Flow in Corrugated Pipes .................. 61
6.2 Simulation Setup .................................................... 62
   6.2.1 Computational Domain Geometry ......................... 63
   6.2.2 Boundary Conditions ....................................... 64
   6.2.3 Meshing, Solution Procedure, Discussion of a Typical Solution . 65
6.3 Influence of Fabric Roughness on the Friction Factor ......... 67

7 Conclusions ............................................................... 71

A Viscosity Adjustment Approach for Roughness Modeling .......... 73
A.1 Approach for Roughness Modeling .............................. 73
A.2 Model Setup in COMSOL Multiphysics™ ...................... 74
A.3 Results ............................................................... 75
A.4 Effects of Mesh Refinement ...................................... 75

B Influence of Wall Region Thickness on the Solution ............. 77
B.1 Model Setup ......................................................... 77
B.2 Results .............................................................. 77
B.3 Conclusions ........................................................ 80

C Performance of Two-Equation Models for Separated Flow Prediction 83
C.1 Performance of the $k - \omega$ Model ............................ 83
C.2 Performance of the $k - \epsilon$ Model ............................ 85
C.3 Discussion of Results ............................................. 87
List of Figures

1.1 Typical flexible pipe. ................................................. 4
1.2 Structure of a flexible pipe: steel spiral wrapped by fabric. .......... 5
1.3 Da Vinci sketch of turbulent flow. ................................... 6
1.4 The Reynolds experiment: (a) laminar flow, (b) early transitional (but still laminar) flow, and (c) turbulence. ............................... 7
1.5 Instantaneous turbulent velocity fields on a plane section of fully developed plane-channel flow at higher (bottom) and lower (top) Reynolds number. DNS by Tellervo Brandt, Helsinki University of Technology / Laboratory for Aerodynamics. ..................................................... 8
3.1 Shear layer. .............................................................. 23
3.2 An example of $v$ versus $u$ (so called quadrant plot) in the wall layer at $y^+ = 19$. Statistical bias towards the 2nd and 4th quadrants is clearly seen. Reproduced from [11]. ............................................................. 26
3.3 Flat-plate boundary layer. ............................................. 28
3.4 Normalized shear stress in a flat-plate turbulent boundary layer. ........ 31
3.5 Sketch showing the various regions of the turbulent boundary layer in inner and outer variables. ........................................ 33
3.6 Velocity profile in a turbulent boundary layer. .......................... 36
4.1 Schematic representation of the mesh for a wall function and a near-wall model approach. .................................................... 40
4.2 Dependence of the friction factor on the thickness of the wall region at Reynolds number of order $10^6$. Solid line - computed values, dotted line - experimental values. .................................................... 44
5.1 Pipe flow (left) and open channel flow (right). ........................ 45
5.2 Flow at the entrance to a pipe. ........................................ 46
5.3 Pressure distribution along the flow in a pipe. ........................... 47
5.4 Moody diagram. ......................................................... 48
5.5 Computational domain ($R = 0.2m$, $L = 0.02$). ....................... 50
5.6 Coarse mesh. .................................................................. 53
5.7 Fine mesh. .................................................................... 53
5.8 Sample solution: velocity distribution. ..................................... 53
5.9 Computed and measured friction factors for smooth pipes ($e/D = 0$). 54
5.10 Computed and measured friction factors for smooth pipes ($e/D = 0$) zoomed around $Re \sim 10^6$. ............................................. 55
5.11 Computed (dotted lines) and measured (solid lines) friction factors for flow with rough walls. .................................................. 56
5.12 Computed (dotted lines) and measured (solid lines) friction factors for flow
with rough walls using a combined law of the wall, Equation 5.33. . . . . . . 58
5.13 Friction factor measured by Nikuradse [17]. . . . . . . . . . . . . . . . . . . 59

6.1 Flexible duct. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 61
6.2 Separation of flow over a curved surface. . . . . . . . . . . . . . . . . . . . . 62
6.3 Computational domain. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 63
6.4 Initial mesh. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 65
6.5 Adapted mesh (generated from the initial mesh by an adaptive solver). . . . 66
6.6 Convergence vs. iteration number. . . . . . . . . . . . . . . . . . . . . . . . 67
6.7 Typical solution. Coloured surface - pressure, arrows - velocity fields, green
   lines - streamlines. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 68
6.8 Pressure profile along the radial direction at $x = 0$. . . . . . . . . . . . . . . 69
6.9 Friction factor vs. fabric roughness. . . . . . . . . . . . . . . . . . . . . . . . 69
A.1 Friction factor computed from simulations. . . . . . . . . . . . . . . . . . . 76

C.1 Computational domain. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 84
C.2 The mesh generated by the adaptive solver around the expansion region. . . 85
C.3 Plot of the solution around the expansion region. Colored surface - $U$ ($x$-
   velocity), black lines - flow streamlines, white line - contourline for $U = 0$. . 86
C.4 The mesh generated by the adaptive solver around the expansion region. . . 87
C.5 Plot of the solution around the expansion region. Colored surface - $U$ ($x$-
   velocity), black lines - flow streamlines, white line - contourline for $U = 0$. . 88
Chapter 1

Introduction

1.1 Goals of the Investigation

Non-metallic flexible pipe products have achieved wide usage in industry. Areas of their applications include heating, ventilation, air-conditioning and, most importantly, connecting terminal or delivery devices to main distribution ducts (such as transport of Liquid Natural Gas from ships to the mainland distribution network).

Flexible ducts are comprised of fabric wrapped over a spiral metal framework. Due to this construction, they respond very well to bending, are cheaper and are much easier to install than metal pipes. Figure 1.1 shows a typical flexible pipe. It is constructed from a neoprene impregnated polyester fabric encapsulating a helix spring of steel wire. This tube has an excellent strength/weight ratio and is able to withstand severe flexing.

The structure of a flexible pipe is seen in Figure 1.2. The steel spiral wire gives strength to the pipe, while the use of fabric instead of a hard material (such as metal) allows for a high degree of flexiblity.

Because of the specific construction, the pipe walls are not straight - they are corrugated. Moreover, the fabric which covers the steel spiral is much rougher than the wall of a metal pipe. This requires more energy (higher pressure difference) to push the flow. Therefore, an important factor in flexible pipe design is to attain minimum pressure loss throughout the distribution line and thus minimize the transportation costs. The pressure loss along a pipe is caused by the friction at the wall (skin friction) and the shape of the wall (drag). The friction is proportional to the pressure loss per unit distance. Therefore, in this investigation, we are interested in estimating the friction factor for turbulent flow at Reynolds number around $10^6$ in a flexible pipe with a specific configuration. In particular, we study the influence of the roughness of the fabric on the friction factor. The performance of different two-equation turbulence models, boundary conditions and computational grids is investigated.

1.2 Descriptions of Turbulence

Turbulence was already recognized as a distinct fluid behavior at least 500 years ago. Figure 1.3 shows a picture of turbulent flow found in a sketch book of Da Vinci, along with a remarkably modern description:

"...the smallest eddies are almost numberless, and large things are rotated only by large eddies and not by small ones, and small things are turned by small eddies and large."

Such phenomena were termed "turbolenza" by Da Vinci, and hence the origin of our
1.2 Descriptions of Turbulence

Figure 1.1: Typical flexible pipe.

modern word for this type of fluid flow [14]. The Navier-Stokes equations, which are now almost universally believed to embody the physics of all fluid flows (within the continuum hypothesis), including turbulent ones, were introduced in the early to mid 19th century by Navier and Stokes. Here we present these in the simple form appropriate for analysis of incompressible flow of a fluid:

\[ \nabla \cdot \mathbf{U} = 0, \]  
\[ \mathbf{U}_t + \mathbf{U} \cdot \nabla \mathbf{U} = \frac{1}{\rho} \nabla p + \nu \Delta \mathbf{U} + \frac{1}{\rho} \mathbf{F}_B. \]

To be more precise, only the second equation, the momentum equation, is usually referred to as Navier-Stokes equation, while the first one is simply called the continuity equation or the mass conservation equation. In these equations, \( \mathbf{U} = (U,V,W)^T \) is the velocity vector which, in general, depends on all three spatial coordinates \( (x,y,z) \), \( p \) is pressure, and \( \mathbf{F}_B \) is a general body-force term. The differential operators \( \nabla \) and \( \Delta \) are the gradient and Laplace operators, respectively, in an appropriate coordinate system. The subscript \( t \) is shorthand notation for time differentiation, \( \partial/\partial t \), \( \nu \) is the kinematic viscosity, and \( \rho \) is the mass density.
These equations are nonlinear and difficult to solve. As is well known, there are few exact solutions, and all of these have been obtained at the expense of introducing simplifying, often physically unrealistic, assumptions. Thus, little progress in the understanding of turbulence can be obtained via analytical solutions to these equations, and as a consequence early descriptions of turbulence were based mainly on experimental observations. O. Reynolds (around 1880) was the first to systematically investigate the transition from laminar to turbulent flow by injecting a dye streak into flow through a pipe having smooth transparent walls. His observations led to identification of a single dimensionless parameter, now called the Reynolds number, and denoted \( Re \),

\[
Re = \frac{UL}{\nu},
\]

that completely characterizes flow behavior in this situation. In this expression \( U \) is a velocity scale (i.e., a "typical" value of velocity, say, the average), and \( L \) is a typical length scale, e.g., the diameter of a pipe through which fluid is flowing. \( Re \) expresses the relative importance of inertial and viscous forces. It is worth noting here that Equation 1.2 can be rescaled and written in terms of \( Re \) as follows:

\[
\dot{U} + U \cdot \nabla U = \nabla p + \frac{1}{Re} \Delta U + \tilde{F}_B.
\]

where pressure has been scaled by twice the dynamic pressure, \( \rho U^2/2 \), and \( \tilde{F}_B \) is a dimensionless body force. One can see from Equation 1.4 that in the absence of body forces \( Re \) is the only free parameter in the Navier-Stokes equations. Hence, setting its value prescribes the solution.

Figure 1.4 provides a sketch of the three flow regimes identified in the Reynolds experiments as \( Re \) is varied. Figure 1.4(a) depicts laminar flow corresponding to \( Re < 2000 \) for which dye injected into the stream can mix with the main flow of water only via molecular diffusion. This process is generally very slow compared with flow speeds, so little mixing, and hence very little apparent spreading of the dye streak, takes place over the length of the tube containing the flowing water. Figure 1.4(b) shows an early transitional state of flow (\( 2000 < Re < 2300 \)) for which the dye streak becomes wavy; but this flow is still laminar as indicated by the fact that the streak is still clearly identifiable with little mixing of dye and water having taken place. Turbulent flow is indicated in Figure 1.4(c). Here we see that instantaneous streamlines (now different from the dye streak itself) change direction erratically, and the dye has mixed significantly with the water. There are a couple of things to note regarding this. First, the enhanced mixing is a very important feature of turbulence, and one that is often sought in engineering processes as in, for example, mixing of reactants in a combustion process, or simply mixing of fluids
1.3 Simulation of Turbulent Flows: DNS, LES and RANS

There are basically three ways to simulate turbulent flow:

- Direct Numerical Simulation (DNS)
- Large-Eddy Simulation (LES)
- Reynolds Averaged Navier-Stokes (RANS)

**Direct Numerical Simulation (DNS)** is a simulation in computational fluid dynamics in which the Navier-Stokes equations are numerically solved without any turbulence model. This means that the whole range of spatial and temporal scales of the turbulence must be resolved. All the spatial scales of the turbulence must be resolved in the computational mesh, from the smallest dissipative scales (Kolmogorov scales), up to the integral scale $L$, associated with the motions containing most of the kinetic energy. Therefore, the computational cost of DNS is very high, even at low Reynolds numbers. For the Reynolds numbers encountered in most industrial applications, the computational resources required by a DNS would exceed the capacity of the most powerful computer.
1.3 Simulation of Turbulent Flows: DNS, LES and RANS

Figure 1.4: The Reynolds experiment: (a) laminar flow, (b) early transitional (but still laminar) flow, and (c) turbulence.

currently available. However, direct numerical simulation is a useful tool in fundamental research in turbulence. Using DNS it is possible to perform "numerical experiments", and extract from them information difficult or impossible to obtain in the laboratory, allowing a better understanding of the physics of turbulence. Also, direct numerical simulations are useful in the development of turbulence models for practical applications, such as sub-grid scale models for Large Eddy Simulation (LES) and models for methods that solve the Reynolds-Averaged Navier-Stokes equations (RANS). The biggest DNS in the world, up to this date, used $4096^3$ mesh points. It was carried out in the Japanese Earth Simulator supercomputer in 2002 [1]. Figure 1.5 shows an instantaneous turbulent velocity field (the color is directly proportional to the magnitude of the velocity vector). The so-called "turbulent eddies" are easily noticeable on the pictures.

Large-Eddy Simulation (LES) is a numerical technique used to solve the partial differential equations governing turbulent fluid flow. A common deduction of Kolmogorov’s (1941) famous theory of self similarity is that large eddies of the flow are dependent on the flow geometry, while smaller eddies are self similar and have a universal character. For this reason, it became a practice to solve only for the large eddies explicitly, and model the effect of the smaller and more universal eddies on the larger ones. Thus, in LES the large scale motions of the flow are calculated, while the effect of the smaller universal scales (the so called sub-grid scales) are modeled using a sub-grid scale (SGS) model. In practical implementations, one is required to solve the filtered Navier-Stokes equations with an additional sub-grid scale stress term. The most commonly used SGS models are
the Smagorinsky model and its dynamic variants. They compensate for the unresolved turbulent scales through the addition of an "eddy viscosity" into the governing equations.

LES requires less computational effort than direct numerical simulation (DNS) but more effort than those methods that solve the Reynolds-averaged Navier-Stokes equations (RANS). The computational demands also increase significantly in the vicinity of walls, and simulating such flows usually exceeds the limits of modern supercomputers today. For this reason, zonal approaches are often adopted, with RANS or other empirically-based models replacing LES in the wall region.

The main advantage of LES over computationally cheaper RANS approaches is the increased level of detail it can deliver. While RANS methods provide "averaged" results, LES is able to predict instantaneous flow characteristics and resolve turbulent flow structures. This is particularly important in simulations involving chemical reactions, such as the combustion of fuel in an engine. While the "averaged" concentration of chemical species may be too low to trigger a reaction, instantaneously there can be localised areas of high concentration in which reactions will occur. LES also offers significantly more accurate results over RANS for flows involving flow separation or acoustic application.

The Reynolds-averaged Navier-Stokes (RANS) equations are time- or ensemble-averaged equations of motion for fluid flow. The averaging process brings new unknown terms into the Navier-Stokes equation. Therefore, additional (closure) equations are needed to be able to solve the system. These equations are derived by taking higher-order moments of the averaged Navier-Stokes equation and making additional assumptions based on the knowledge of the properties of the turbulent flow. This process (detailed in Chapter 2) results in a modified set of equations that is computationally less expensive to solve. Given our purposes and the available computational power, we will use RANS models (specifically $k$ – $\epsilon$ and, to a lesser extent, $k$ – $\omega$) for our simulations.
1.4 Report Outline

This report contains seven chapters. In Chapter 2 the derivation of the $k - \epsilon$ and $k - \omega$ models is given. Starting from the Navier-Stokes equations for incompressible flow, the Reynolds averaged equations are derived. Then, by taking higher moments of the latter and making certain assumptions, two additional equations are derived. The result of these manipulations is a closed system of six equations with six unknowns.

Chapter 3 discusses turbulent boundary layers. First, an order-of-magnitude estimate is performed for the free shear layer equations. Then, a wall-bounded shear layer is considered and the need for a division of the boundary layer into an "inner" and an "outer" layer is explained. Finally, the so-called law of the wall is derived, which gives the velocity profile in the region of overlap of the "inner" and "outer" layers.

Chapter 4 gives a detailed explanation of the use of the law of the wall as a boundary condition. First, two approaches of near-wall modeling are presented. Then, specific boundary conditions for the velocity and the turbulent variables are derived.

Chapter 5 starts with a brief review on turbulent flow in pipes. Then a simulation of turbulent flow in a non-corrugated pipe is set up and all the characteristics of the model (computational domain, boundary conditions, mesh, etc.) are described in the process. A series of simulations is performed to validate the model. Two cases are validated: flow through a pipe with smooth walls and flow through a pipe with rough walls. The modifications made to the boundary conditions to account for wall roughness are described in detail.

In Chapter 6 the computational results for the corrugated pipe are presented. The simulation results are used to estimate the friction factor for different degrees of wall roughness.

At the end, Chapter 7 summarizes the results of this study and suggests directions for future work.
Chapter 2

Two-Equation Turbulence Models

Turbulent flow is characterized by small fluctuations of the physical quantities such as velocity or pressure. They cause instability when trying to solve the original flow equations numerically and require extremely fine meshes and small timesteps. To overcome these difficulties, it is proposed to average the equations so that only the evolution of average quantities is described. The averaging process brings new unknown terms into the Navier-Stokes equation. In order to close the model, additional equations are derived by taking higher-order moments of the averaged Navier-Stokes equation and making additional assumptions based on the knowledge of the properties of the turbulent flow. This process results in a set of six equations with six unknowns.

2.1 Equations Describing the Dynamics of Flow

All fluid flows (laminar as well as turbulent) are governed by the Navier-Stokes (momentum) equations and continuity equation. Using the tensor notation, they are written as (neglecting body forces, such as gravity):

\[
\begin{align*}
\frac{\partial (\tilde{\rho} \tilde{u}_i)}{\partial t} + \tilde{u}_j \frac{\partial (\tilde{\rho} \tilde{u}_i)}{\partial x_j} &= -\frac{\partial \tilde{p}}{\partial x_i} + \frac{\partial \tilde{T}^{(v)}_{ij}}{\partial x_j}, \\
\frac{\partial \tilde{\rho}}{\partial t} + \tilde{u}_j \frac{\partial \tilde{\rho}}{\partial x_j} + \tilde{\rho} \frac{\partial \tilde{u}_j}{\partial x_j} &= 0,
\end{align*}
\]

where \( \tilde{u}_i(\vec{x}, t) \) represents the \( i \)-th component of the fluid velocity at a point in space, \( [\vec{x}]_i = x_i \), and time, \( t \). Also \( \tilde{p}(\vec{x}, t) \) represents the pressure, \( \tilde{T}^{(v)}_{ij}(\vec{x}, t) \), the viscous stresses, and \( \tilde{\rho} \) the fluid density. The tilde over the symbol indicates that an instantaneous quantity is being considered. To write the equations in a concise form, the Einstein summation convention has been used.

In Equation 2.1, the subscript \( i \) is a free index which (in 3D) can take the values 1, 2 and 3. Thus Equation 2.1 actually represents three separate equations. These three equations are just Newton’s second law written for a continuum in a spatial (or Eulerian) reference frame (conservation of momentum). Together they relate the rate of change of momentum per unit mass (\( \tilde{\rho} \tilde{u}_i \)), a vector quantity, to the contact forces (viscous stresses) and body forces (pressure).

Equation 2.2 is the equation for mass conservation in the absence of sources (or sinks) of mass. The flow considered in this report (such as LNG flow) is assumed to be incom-
pressible, which implies that the derivative of the density following the fluid material (the term in brackets) is zero. Thus for incompressible flows

$$\frac{D\tilde{\rho}}{Dt} = \frac{\partial \tilde{\rho}}{\partial t} + \tilde{u}_j \frac{\partial \tilde{\rho}}{\partial x_j} = 0$$  \hspace{1cm} (2.3)$$

and from Equation 2.2 it follows that the mass conservation equation reduces to

$$\frac{\partial \tilde{u}_j}{\partial x_j} = 0. \hspace{1cm} (2.4)$$

The viscous stresses (the stress minus the mean normal stress, which is the pressure) are represented by the tensor $\tilde{T}_{ij}^{(v)}$. From its definition, $\tilde{T}_{kk}^{(v)} = 0$. In many flows of interest, the fluid behaves as a Newtonian fluid in which the viscous stress can be related to the fluid motion by a constitutive relation of the form:

$$\tilde{T}_{ij}^{(v)} = 2\mu [\tilde{s}_{ij} - \frac{1}{3} \tilde{s}_{kk} \delta_{ij}]. \hspace{1cm} (2.5)$$

The (dynamic) viscosity, $\mu$, is a property of the fluid that can be measured in an independent experiment. $\tilde{s}_{ij}$ is the instantaneous strain rate tensor defined by

$$\tilde{s}_{ij} = \frac{1}{2} \left[ \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right]. \hspace{1cm} (2.6)$$

From its definition, $\tilde{s}_{kk} = \frac{\partial \tilde{u}_k}{\partial x_k}$. If the flow is incompressible, $\tilde{s}_{kk} = 0$ and the Newtonian constitutive equation reduces to

$$\tilde{T}_{ij}^{(v)} = 2\mu \tilde{s}_{ij}, \Rightarrow \frac{\partial \tilde{T}_{ij}^{(v)}}{\partial x_j} = 2\mu \frac{1}{2} \left[ \frac{\partial^2 \tilde{u}_i}{\partial x_j^2} + \frac{\partial \tilde{u}_j}{\partial x_i} \frac{\partial \tilde{u}_j}{\partial x_j} \right] = \mu \frac{\partial^2 \tilde{u}_i}{\partial x_j^2}. \hspace{1cm} (2.7)$$

Throughout this report, the density $\tilde{\rho} = \rho$ and the viscosity $\mu$ will be assumed constant. With these assumptions, the instantaneous momentum equations for a Newtonian Fluid reduce to:

$$\rho \left[ \frac{\partial \tilde{u}_i}{\partial t} + \tilde{u}_j \frac{\partial \tilde{u}_i}{\partial x_j} \right] = -\frac{\partial \tilde{p}}{\partial x_i} + \mu \frac{\partial^2 \tilde{u}_i}{\partial x_j^2}. \hspace{1cm} (2.8)$$

Note that since the density is assumed constant, the tilde over $\rho$ is no longer necessary. Sometimes it will be more convenient to not explicitly include incompressibility in the stress term, but to refer to the incompressible momentum equation in the following form:

$$\rho \left[ \frac{\partial \tilde{u}_i}{\partial t} + \tilde{u}_j \frac{\partial \tilde{u}_i}{\partial x_j} \right] = -\frac{\partial \tilde{p}}{\partial x_i} + \frac{\partial \tilde{T}_{ij}^{(v)}}{\partial x_j}. \hspace{1cm} (2.9)$$

This form has the advantage that it is easier to keep track of the exact role of the viscous stresses.

### 2.2 Reynolds Averaging

Although laminar flow solutions to the equations often exist that are consistent with the boundary conditions, perturbations to these solutions (sometimes even infinitesimal) can cause them to become turbulent [9]. To see how this can happen, it is convenient to analyze
2.2 Reynolds Averaging

The flow in two parts, a mean (or average) component and a fluctuating component. Thus the instantaneous velocity and stresses can be written as:

\[
\begin{align*}
\tilde{u}_i &= U_i + u_i, \\
\tilde{p} &= P + p, \\
\tilde{T}^{(v)}_{ij} &= T^{(v)}_{ij} + \tau^{(v)}_{ij},
\end{align*}
\]

where \(U_i, P\) and \(T^{(v)}_{ij}\) represent the mean components, and \(u_i, p\) and \(\tau^{(v)}_{ij}\) the fluctuating components. This technique for decomposing the instantaneous motion is referred to as the Reynolds decomposition [21].

Substitution of Equations 2.10-2.12 into Equation 2.9 yields

\[
\rho \left[ \frac{\partial (U_i + u_i)}{\partial t} + (U_j + u_j) \frac{\partial (U_i + u_i)}{\partial x_j} \right] = - \frac{\partial (P + p)}{\partial x_i} + \frac{\partial (T^{(v)}_{ij} + \tau^{(v)}_{ij})}{\partial x_j}.
\]

(2.13)

This equation can now be averaged in time to yield an equation expressing momentum conservation for the averaged motion. Due to linearity of the differential operator, the operations of averaging and differentiation commute (i.e., the average of a derivative is the same as the derivative of the average). Also the average of a fluctuating quantity is zero. Thus, the equation for the averaged motion reduces to:

\[
\rho \left[ \frac{\partial \tilde{U}_i}{\partial t} + U_j \frac{\partial \tilde{U}_i}{\partial x_j} \right] = - \frac{\partial \tilde{P}}{\partial x_i} + \frac{\partial \tilde{T}^{(v)}_{ij} + \tilde{\tau}^{(v)}_{ij}}{\partial x_j} - \rho \left< u_j \frac{\partial u_i}{\partial x_j} \right>,
\]

(2.14)

where the remaining fluctuating product term has been moved to the right hand side of the equation. The angle brackets are used to denote the time average of a quantity. Whether or not the last term is zero like the other fluctuating term depends on the correlation of the terms in the product. In general, these correlations are not zero.

The mass conservation equation can be similarly decomposed. In incompressible form, substitution of Equation 2.10 into Equation 2.4 yields:

\[
\frac{\partial (U_j + u_j)}{\partial x_j} = 0,
\]

(2.15)

of which the average is

\[
\frac{\partial \tilde{U}_j}{\partial x_j} = 0.
\]

(2.16)

It is clear from the Equation 2.16 that the averaged motion satisfies the same form of the mass conservation equation as does the instantaneous motion, at least for incompressible flows. Unfortunately, as is easily seen from Equation 2.14, this is not true for the momentum conservation.

Equation 2.16 can be subtracted from Equation 2.15 to yield an equation for fluctuating motion alone; i.e,

\[
\frac{\partial u_j}{\partial x_j} = 0.
\]

(2.17)
Again, like for the mean flow, the form of the original instantaneous equation is seen to be preserved for the turbulent fluctuations. The reason is obvious: the continuity equation is linear. The momentum equation, on the other hand, is not; hence the difference.

Equation 2.17 can be used to rewrite the last term in Equation 2.14 for the mean momentum. Multiplying Equation 2.17 by $u_i$ and averaging yields:

$$\langle u_i \frac{\partial u_j}{\partial x_j} \rangle = 0.$$  \hspace{1cm} (2.18)

This can be added to $\langle u_j \frac{\partial u_i}{\partial x_j} \rangle$ to obtain:

$$\langle u_j \frac{\partial u_i}{\partial x_j} \rangle + 0 = \langle u_j \frac{\partial u_i}{\partial x_j} \rangle + \langle u_i \frac{\partial u_j}{\partial x_j} \rangle = \frac{\partial}{\partial x_j} \langle u_i u_j \rangle,$$  \hspace{1cm} (2.19)

where, again, has been used the fact that arithmetic and averaging operations commute.

The equation for the averaged momentum, Equation 2.14 can now be rewritten as:

$$\rho \left[ \frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} \right] = -\frac{\partial P}{\partial x_i} + \frac{\partial T^{(v)}_{ij}}{\partial x_j} - \frac{\partial}{\partial x_j} (\rho \langle u_i u_j \rangle).$$  \hspace{1cm} (2.20)

The last two terms on the right hand side are both divergence terms and can be combined; the result is:

$$\rho \left[ \frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} \right] = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} [T^{(v)}_{ij} - \rho \langle u_i u_j \rangle].$$  \hspace{1cm} (2.21)

Now the terms in square brackets on the right have the dimensions of stress. The first term is, in fact, the viscous (molecular) stress. The second term, on the other hand, is not a stress at all, but simply a re-worked version of the fluctuating contribution to the non-linear acceleration terms. The fact that it can be written this way, however, indicates that at least as far as the motion is concerned, it acts as though it were a stress; hence its name, the Reynolds stress (or turbulence stress).

### 2.3 Eddy Viscosity Approximation

From the point of view of the averaged motion, at least, the problem with the non-linearity of the instaneous equations is that they introduce new unknowns, the Reynolds stress into the averaged equations. There are six individual stress components we must deal with: $\langle u_1^2 \rangle$, $\langle u_2^2 \rangle$, $\langle u_3^2 \rangle$, $\langle u_1 u_2 \rangle$, $\langle u_1 u_3 \rangle$, and $\langle u_2 u_3 \rangle$. These have to be related to the mean motion itself before the equations can be solved, since the number of unknowns and number of equations must be equal. The absence of these additional equations is often referred to as the closure problem of turbulence modeling.

This problem arose also when the instantaneous equations were written (Equations 2.1 and 2.2), since relations had to be introduced to relate the stresses to the motion itself. These relations (or constitutive equations) depended only on the properties of the fluid material, and not on the flow itself. Because of this fact, it is possible to carry out independent experiments, called viscometric experiments, in which these fluid properties can be determined once and for all. Equation 2.5 provides an example just such a constitutive relation, the viscosity, $\mu$, depending only in the choice of the material. For example, once the viscosity of water at given temperature is determined, this value can be used in all flows at that temperature, not just the one in which the evaluation was made.
2.4 The Reynolds Stress Equation

We can try such an approach for the turbulence Reynolds stresses. For example, a Newtonian type closure for the Reynolds stresses, often referred to as an "eddy" or "turbulent" viscosity model (sometimes also as Boussinesq approximation in the honor of Joseph Valentin Boussinesq who suggested using this closure equation [6]), looks like:

\[- \rho \langle u_i u_j \rangle = 2 \mu_T \left[ S_{ij} - \frac{1}{3} S_{kk} \delta_{ij} \right], \tag{2.22}\]

where \( \mu_T \) is the turbulence "viscosity" (also called the eddy viscosity) and \( S_{ij} \) is the mean strain rate defined by:

\[ S_{ij} = \frac{1}{2} \left[ \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right]. \tag{2.23}\]

The second term vanishes identically for incompressible flow. Therefore, Equation 2.22 reduces to

\[- \rho \langle u_i u_j \rangle = \mu_T \left[ \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right]. \tag{2.24}\]

However, experiments show that \( \mu_T \) has different values for different flows (even though the fluid used is the same). This is explained by the fact that turbulence is a property of the flow, not of the fluid. Following these conclusions, in two-equation models an equation is developed which relates \( \mu_T \) to the properties of the turbulent flow (such as turbulent kinetic energy and turbulence dissipation).

2.4 The Reynolds Stress Equation

We can try to find an equation for the unknown Reynolds stress by taking moments of the Navier-Stokes equation, as described in [27]. That is, we multiply the Navier-Stokes equation by the fluctuating velocity and time average the product. Using this procedure we can derive a partial differential equation for the Reynolds-stress tensor. To illustrate the process, a special notation is introduced. Let \( N(\tilde{u}_i) \) denote the Navier-Stokes operator as used in Equation 2.8, then

\[ N(\tilde{u}_i) = \rho \frac{\partial \tilde{u}_i}{\partial t} + \rho \tilde{u}_k \frac{\partial \tilde{u}_i}{\partial x_k} + \frac{\partial \tilde{p}}{\partial x_i} - \mu \frac{\partial^2 \tilde{u}_i}{\partial x_k^2}, \tag{2.25}\]

or, by using the tensor notation for derivatives

\[ N(\tilde{u}_i) = \rho \tilde{u}_{i,t} + \rho \tilde{u}_{k,i,k} + \tilde{p}_{,i} - \mu \tilde{u}_{i,kk}. \tag{2.26}\]

Thus, the Navier-Stokes equation can be written symbolically as

\[ N(\tilde{u}_i) = 0. \tag{2.27}\]

In order to derive an equation for the Reynolds stress tensor, the following time average is formed:

\[ \langle u_i N(\tilde{u}_j) + u_j N(\tilde{u}_i) \rangle = 0. \tag{2.28}\]

\footnote{\textsuperscript{1}Joseph Valentin Boussinesq (born March 13, 1842 in Saint-André-de-Sangonis (Hérault département), died February 19, 1929 in Paris) was a French mathematician and physicist who made significant contributions to the theory of hydrodynamics, vibration, light, and heat.}
2.4 The Reynolds Stress Equation

The above equation is symmetric in $i$ and $j$, as we expect it to be, because of the symmetry of the Reynolds stress tensor. To keep the calculations simple, we proceed term by term. First we consider the unsteady term:

$$
\langle u_i (\rho \tilde{u}_j)_t + u_j (\rho \tilde{u}_i)_t \rangle = \langle \rho u_i (U_j + u_j)_t \rangle + \langle \rho u_j (U_i + u_i)_t \rangle
$$

$$
= \langle \rho u_i U_j,t \rangle + \langle \rho u_i u_j,t \rangle + \langle \rho u_j U_i,t \rangle + \langle \rho u_j u_i,t \rangle
$$

$$
= \langle \rho u_i u_j,t \rangle + \langle \rho u_j u_i,t \rangle
$$

$$
= \rho \langle (u_i u_j)_t \rangle = -\frac{\partial \sigma_{ij}}{\partial t}.
$$

(2.29)

where in the last row we used the following notation for the Reynolds stress

$$
\sigma_{ij} = -\rho \langle u_i u_j \rangle.
$$

(2.30)

Turning to the convective term, we have

$$
\langle \rho u_i \tilde{u}_k \tilde{u}_j,k + \rho u_j \tilde{u}_k \tilde{u}_i,k \rangle = \langle \rho u_i (U_k + u_k)(U_j + u_j)_k \rangle + \langle \rho u_j (U_k + u_k)(U_i + u_i)_k \rangle
$$

$$
= \langle \rho u_i U_k U_j,k \rangle + \langle \rho u_i u_k (U_j + u_j)_k \rangle + \langle \rho u_j u_k (U_i + u_i)_k \rangle
$$

$$
= \rho U_k \langle u_i u_j \rangle,k + \rho (u_k (u_i u_j)_k) + \rho (u_i u_k) U_j,k + \rho (u_j u_k) U_i,k
$$

$$
= \rho U_k \langle u_i u_j \rangle,k + \rho (u_k (u_i u_j)_k) + \rho (u_i u_k) U_j,k + \rho (u_j u_k) U_i,k
$$

$$
= \rho U_k \langle u_i u_j \rangle,k + \rho (u_i u_k) U_j,k + \rho (u_j u_k) U_i,k
$$

$$
= \rho U_k \langle u_i u_j \rangle,k + \rho (u_i u_k) U_j,k + \rho (u_j u_k) U_i,k
$$

$$
= -U_k \frac{\partial \sigma_{ij}}{\partial x_k} - \sigma_{ik} \frac{\partial U_j}{\partial x_k} - \sigma_{jk} \frac{\partial U_i}{\partial x_k} + \rho \frac{\partial }{\partial x_k} (u_i u_j u_k).
$$

(2.31)

We are allowed to add the term $\rho (u_k,k (u_i u_j))$ in the seventh row, because $u_{k,k} = \frac{\partial u_k}{\partial x_k} = 0$ (continuity equation for fluctuations, see Equation 2.17).

The pressure gradient term is straightforward:

$$
\langle u_i \tilde{p}_j + u_j \tilde{p},i \rangle = \langle u_i (P + p),j \rangle + \langle u_j (P + p),i \rangle
$$

$$
= \langle u_i p_j \rangle + \langle u_j p,i \rangle
$$

$$
= \langle u_i \frac{\partial p}{\partial x_j} + u_j \frac{\partial p}{\partial x_i} \rangle.
$$

(2.32)

Finally, the viscous term yields
\[ \mu (u_i \tilde{u}_{j,kk} + u_j \tilde{u}_{i,kk}) = \mu (u_i (U_j + u_j)_{kk}) + \mu (u_j (U_i + u_i)_{kk}) \]
\[ = \mu (u_i u_{j,kk}) + \mu (u_j u_{i,kk}) \]
\[ = \mu (u_i u_{j,k}) + \mu (u_j u_{i,k}) - 2\mu (u_{j,k} u_{i,k}) \]
\[ = -\mu \frac{\partial^2 \sigma_{ij}}{\rho \partial x_k^2} - 2\mu \left( \frac{\partial u_j}{\partial x_k} \frac{\partial u_i}{\partial x_k} \right). \] (2.33)

Collecting the terms, we arrive at the PDE for the Reynolds stress tensor:
\[ \frac{\partial \sigma_{ij}}{\partial t} + U_k \frac{\partial \sigma_{ij}}{\partial x_k} = -\sigma_{ik} \frac{\partial U_j}{\partial x_k} - \sigma_{jk} \frac{\partial U_i}{\partial x_k} + 2\mu \left( \frac{\partial u_j}{\partial x_k} \frac{\partial u_i}{\partial x_k} \right) + \left( u_i \frac{\partial p}{\partial x_j} + u_j \frac{\partial p}{\partial x_i} \right) \]
\[ + \frac{\partial}{\partial x_k} \left[ \frac{\mu}{\rho} \frac{\partial \sigma_{ij}}{\partial x_k} + \rho (u_i u_j u_k) \right]. \] (2.34)

This is known as the Reynolds-stress equation. It supplies nine new relations for the components of the Reynolds-stress tensor. Out of them, six equations are independent and three are redundant because of the symmetry of the Reynolds-stress tensor. At the same time, we have also generated a much bigger number of unknowns with terms like
\[ \langle u_i u_j u_k \rangle - 27 \text{ dependent unknowns} \]
\[ 2\mu \langle \frac{\partial u_j}{\partial x_k} \frac{\partial u_i}{\partial x_k} \rangle - 27 \text{ dependent unknowns} \]
\[ \langle u_i \frac{\partial p}{\partial x_j} + u_j \frac{\partial p}{\partial x_i} \rangle - 9 \text{ dependent unknowns} \]

Again, we are confronted with the closure problem of turbulence modeling. Because of the nonlinearity of the equation for momentum conservation, as we take higher and higher moments (multiply with a fluctuating quantity and time average), we generate an additional number of equations and unknowns. Unfortunately, the number of unknowns is increasing much faster than the number of equations. From the physics point of view, this situation is to be expected, because by taking higher moments we are performing a strictly mathematical operation, without adding any new physical principles (laws). In the next sections we will look for approximations for the unknown correlations in terms of flow properties that are known such that a sufficient number of equations exists. By making appropriate approximations the systems is closed.

By a rearrangement of terms, the Reynolds-stress Equation 2.34 can be cast into a form which has gained a wider use:
\[ \frac{\partial \sigma_{ij}}{\partial t} + U_k \frac{\partial \sigma_{ij}}{\partial x_k} = -\sigma_{ik} \frac{\partial U_j}{\partial x_k} - \sigma_{jk} \frac{\partial U_i}{\partial x_k} + \rho \epsilon_{ij} - \Pi_{ij} \]
\[ + \frac{\partial}{\partial x_k} \left[ \frac{\mu}{\rho} \frac{\partial \sigma_{ij}}{\partial x_k} + C_{ijk} \right], \] (2.35)

where
2.5 The Turbulence Energy Equation

The turbulence energy equation incorporates nonlocal and flow history effects. The kinetic energy per unit mass of the turbulent fluctuations, \( k \), is defined as

\[
k = \frac{1}{2} \left( \langle u_i^2 \rangle + \langle u_j^2 \rangle + \langle u_k^2 \rangle \right).
\]

(2.39)

The quantity \( k \) should strictly be referred to as specific turbulence kinetic energy (“specific” meaning “per unit mass”), but is often just called turbulence kinetic energy.

In the previous section we derived the equation for the Reynolds stress tensor. The connection between the Reynolds stress tensor and the turbulent kinetic energy is obtained by taking the trace of the former.

\[
\sigma_{ii} = -\rho \langle u_i u_i \rangle = -2\rho k.
\]

(2.40)

Thus, the trace of the Reynolds stress tensor is proportional to the kinetic energy per unit mass of the turbulent fluctuations. This suggests that we can obtain the equation for the turbulence kinetic energy, \( k \), just by taking the trace of the Reynolds stress equation. In other words we sum up the equations for the stresses located on the diagonal of the second order tensor.

We start from (the expanded version of) Equation 2.35.

\[
\frac{\partial \sigma_{ij}}{\partial t} + U_k \frac{\partial \sigma_{ij}}{\partial x_k} = -\sigma_{ik} \frac{\partial U_i}{\partial x_k} - \sigma_{jk} \frac{\partial U_j}{\partial x_k} + 2\mu \langle \frac{\partial u_j}{\partial x_i} \frac{\partial u_i}{\partial x_k} \rangle - \langle p \frac{\partial u_j}{\partial x_i} + p \frac{\partial u_i}{\partial x_j} \rangle + \frac{\partial}{\partial x_k} \left[ \frac{\mu}{\rho} \frac{\partial \sigma_{ij}}{\partial x_k} + \rho \langle u_i u_j u_k \rangle + \langle pu_i \rangle \delta_{jk} + \langle pu_j \rangle \delta_{ik} \right].
\]

(2.41)

Now we add the equations for \( \sigma_{11}, \sigma_{22}, \) and \( \sigma_{33}, \) to obtain

\[
\frac{\partial \sigma_{ii}}{\partial t} + U_k \frac{\partial \sigma_{ii}}{\partial x_k} = -\sigma_{ik} \frac{\partial U_i}{\partial x_k} - \sigma_{ik} \frac{\partial U_i}{\partial x_k} + 2\mu \langle \frac{\partial u_i}{\partial x_i} \frac{\partial u_i}{\partial x_k} \rangle - \langle p \frac{\partial u_i}{\partial x_i} + p \frac{\partial u_i}{\partial x_i} \rangle + \frac{\partial}{\partial x_k} \left[ \frac{\mu}{\rho} \frac{\partial \sigma_{ii}}{\partial x_k} + \rho \langle u_i u_i u_k \rangle + \langle pu_i \rangle \delta_{ik} + \langle pu_i \rangle \delta_{ik} \right].
\]

(2.42)

Using Equation 2.40 and the fact that \( \langle p \frac{\partial u_i}{\partial x_i} + p \frac{\partial u_i}{\partial x_i} \rangle = 0 \) (because of continuity, Equation 2.17), we have

\[
-2\rho \frac{\partial k}{\partial t} - 2\rho U_k \frac{\partial k}{\partial x_k} = -2\sigma_{ik} \frac{\partial U_i}{\partial x_k} + 2\mu \langle \frac{\partial u_i}{\partial x_k} \frac{\partial u_i}{\partial x_k} \rangle + \frac{\partial}{\partial x_k} \left[ -2\mu \frac{\partial k}{\partial x_k} + \rho \langle u_i u_i u_k \rangle + 2\langle pu_i \rangle \delta_{ik} \right].
\]

(2.43)
The Turbulence Energy Equation

\[ \rho \frac{\partial k}{\partial t} + \rho U_k \frac{\partial k}{\partial x_k} = \sigma_{ik} \frac{\partial U_i}{\partial x_k} - \mu \left( \frac{\partial u_i}{\partial x_k} \frac{\partial u_i}{\partial x_k} \right) + \frac{\partial}{\partial x_k} \left[ \mu \frac{\partial k}{\partial x_k} - \frac{1}{2} \rho \langle u_i u_i u_k \rangle - \langle pu_i \rangle \delta_{ik} \right]. \] (2.44)

The last term is transformed as follows, \( \langle pu_i \rangle \delta_{ik} = \langle pu_k \rangle \), and for the sake of notation, the index \( k \) is replaced by \( j \).

\[ \rho \frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial x_j} = P_k - \rho \epsilon \]
\[ + \frac{\partial}{\partial x_j} \left[ \mu \frac{\partial k}{\partial x_j} - \frac{1}{2} \rho \langle u_i u_i u_j \rangle - \langle pu_j \rangle \right]. \] (2.45)

where \( \epsilon \) is the dissipation per unit mass defined as

\[ \epsilon = \mu \langle \frac{\partial u_i}{\partial x_j} \frac{\partial u_i}{\partial x_j} \rangle, \] (2.46)

and \( P_k \) is the production of turbulent kinetic energy defined as

\[ P_k = \sigma_{ij} \frac{\partial U_i}{\partial x_j} = -\rho < u_i u_j > \frac{\partial U_i}{\partial x_j}. \] (2.47)

The various terms appearing in Equation 2.45 represent physical processes occurring as the turbulence evolves in a given flow. The sum of the two terms on the LHS, the unsteady term and the advection, is the material (Lagrangian) derivative of \( k \) that gives the rate of change of \( k \) following a fluid particle (in a mean-flow, fluctuations are neglected). The Production term represents the rate at which kinetic energy is transferred from the mean flow to the turbulence (to fluctuations). Rewritten as \( \sigma_{ij} S_{ij} \) this term is seen to be the rate at which work is done by the mean strain rate against the turbulent stresses. Dissipation is the rate at which turbulence kinetic energy is converted into thermal internal energy, equal to the mean rate at which work is done by the fluctuating part of the strain rate against the fluctuating viscous stresses. The term involving \( \mu \partial k/\partial x_j \) is called Molecular Diffusion, and represents the diffusion of turbulence energy caused by fluid’s natural molecular transport process. The triple velocity correlation term is referred to as Turbulent Transport, and is regarded as the rate at which turbulence energy is transported through the fluid by turbulent fluctuations. The last term on the RHS of the equation is called Pressure Diffusion, another form of turbulent transport resulting from correlation of pressure and velocity fluctuations.

The standard approximation made to the last two terms of Equation 2.45 is given by:

\[ \frac{1}{2} \rho \langle u_i u_i u_j \rangle + \langle pu_j \rangle = -\frac{\mu_T}{\sigma_k} \frac{\partial k}{\partial x_j}. \] (2.48)

where \( \sigma_k \) is a closure coefficient and \( \mu_T \) is the turbulent (or eddy) viscosity (which comes from the Boussinesq approximation, Equation 2.22). For now, Equation 2.48 simply defines \( \sigma_k \). This statement applies to all turbulence closure coefficients to be used. At this point,
no approximation has been made, although it is hoped that the model is realistic enough that $\sigma_k$ can be chosen to be constant.

Substituting Equation 2.48 in Equation 2.45 we can write the modeled version of the turbulence kinetic energy equation that is used in virtually all turbulence energy equation models. The equation assumes the following form:

$$\frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial x_j} = \sigma_{ij} \frac{\partial U_i}{\partial x_j} - \rho \epsilon + \frac{\partial}{\partial x_j} \left[ (\mu + \mu_T \sigma_k) \frac{\partial k}{\partial x_j} \right]. \quad (2.49)$$

### 2.6 The Turbulence Dissipation Equation and the $k-\epsilon$ Model

In formulating the $k-\epsilon$ model, the idea is to derive the exact equation for $\epsilon$ and from it to find suitable approximations for the exact equation governing its behaviour. The turbulence dissipation, $\epsilon$, is defined in Equation 2.46. The exact equation for $\epsilon$ is derived by taking the following moment of the Navier-Stokes equation:

$$2 \mu \left\langle \frac{\partial u_i}{\partial x_j} \partial N(u_i) \right\rangle = 0, \quad (2.50)$$

where $N(u_i)$ is the Navier-Stokes operator defined in Equation 2.25.

After a considerable amount of algebra, the following equation for $\epsilon$ results:

$$\rho \frac{\partial \epsilon}{\partial t} + \rho U_j \frac{\partial \epsilon}{\partial x_j} = -2 \mu \left[ \langle u_i, k u_j, k \rangle \right] \frac{\partial U_i}{\partial x_j} - 2 \mu \langle u_k u_i, j \rangle \frac{\partial^2 U_i}{\partial x_k \partial x_j} - 2 \mu \langle u_i, k u_i, m \rangle - 2 \mu \langle u_i, u_i, m \rangle - 2 \rho \frac{\partial}{\partial x_j} \left[ \left( \mu + \rho_T \sigma_k \right) \frac{\partial \epsilon}{\partial x_j} \right]. \quad (2.51)$$

This equation is far more complicated than the turbulence kinetic energy equation and involves several new unknown double and triple correlations of fluctuating velocities, pressures and velocity gradients. These correlations are essentially impossible to measure, which provides little hope of finding reliable guidance from experimentalists regarding suitable closure approximations. Therefore, dimensional-analysis arguments are usually used to arrive at the Turbulence Dissipation Equation used in practice:

$$\rho \frac{\partial \epsilon}{\partial t} + \rho U_j \frac{\partial \epsilon}{\partial x_j} = C_{\epsilon 1} \frac{\epsilon}{k} \sigma_{ij} \frac{\partial U_i}{\partial x_j} - C_{\epsilon 2} \rho \frac{\epsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \rho_T \sigma_k \right) \frac{\partial \epsilon}{\partial x_j} \right]. \quad (2.52)$$

where $C_{\epsilon 1}$, $C_{\epsilon 2}$ and $\sigma_\epsilon$ are additional constants.

With the equation above, we have all the necessary information to write the final form of the standart $k-\epsilon$ model.

**Mean flow equations** Mass conservation (Equation 2.16):

$$\frac{\partial U_j}{\partial x_j} = 0. \quad (2.53)$$

Momentum conservation (from Equations 2.14, 2.19, 2.30):

$$\rho \left[ \frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} \right] = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ T_{ij}^{(e)} + \sigma_{ij} \right], \quad (2.54)$$
where (from Equations 2.5, 2.6, and 2.24)

$$T_{ij}^{(v)} = \mu \left[ \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right], \quad \sigma_{ij} = \mu_T \left[ \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right]. \quad (2.55)$$

**Transport equations for standard \( k-\epsilon \) model**

**Turbulent energy equation** (Equation 2.49):

$$\rho \frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial x_j} = \sigma_{ij} \frac{\partial U_i}{\partial x_j} - \rho \epsilon + \frac{\partial}{\partial x_j} \left[ (\mu + \mu_T) \frac{\partial k}{\partial x_j} \right]. \quad (2.56)$$

**Turbulent dissipation equation** (Equation 2.52):

$$\rho \frac{\partial \epsilon}{\partial t} + \rho U_j \frac{\partial \epsilon}{\partial x_j} = C_{\epsilon 1} \epsilon \sigma_{ij} \frac{\partial U_i}{\partial x_j} - C_{\epsilon 2} \frac{\epsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_T \right) \frac{\partial \epsilon}{\partial x_j} \right]. \quad (2.57)$$

Turbulent viscosity is modeled (on dimensional grounds) as

$$\mu_T = \rho C_{\mu} \frac{k^2}{\epsilon}, \quad (2.58)$$

with \( C_{\mu} \) being another constant.

**Model constants** (determined empirically or by fitting to experimental results)

$$C_{\epsilon 1} = 1.44, \quad C_{\epsilon 2} = 1.92, \quad C_{\mu} = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\epsilon = 1.3. \quad (2.59)$$

Thus, we have six equations for six unknowns \((U_1, U_2, U_3, P, k, \epsilon)\), i.e. we achieved closure.

### 2.7 The \( \omega \)-Equation and the \( k-\omega \) Model

The quantity \( \omega \) is interpreted as the ratio of \( \epsilon \) to \( k \), i.e., the rate of dissipation per unit turbulence kinetic energy. Combining physical reasoning with dimensional analysis, Kolmogorov postulated in his 1942 paper [13] the following equation for \( \omega \):

$$\rho \frac{\partial \omega}{\partial t} + \rho U_j \frac{\partial \omega}{\partial x_j} = -\beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ (\sigma_\omega \mu_T) \frac{\partial \omega}{\partial x_j} \right]. \quad (2.60)$$

The form of the equation for \( \omega \) has changed as the \( k-\omega \) model has evolved over the past six decades. A production term has been added to the original Kolmogorov’s equation. Like Kolmogorov, Wilcox in [26] writes the equation for \( \omega \) in terms of \( \omega \). By contrast, most other \( k-\omega \) models use an equation for \( \omega^2 \). Because it has been tested more extensively than any other model, Wilcox’s \( k-\omega \) model [26] is considered to be a state-of-the-art formulation. Moreover, this very model is the one implemented in most computational packages (including Comsol Multiphysics).

Below are given the equations of Wilcox’s \( k-\omega \) model. The mean flow is again described by Equations 2.53-2.55.

**Transport equations for standard \( k-\omega \) model**

**Turbulent energy equation**:
2.7 The $\omega$-Equation and the $k - \omega$ Model

\[
\frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial x_j} = \sigma_{ij} \frac{\partial U_i}{\partial x_j} - \beta \rho k \omega + \frac{\partial}{\partial x_j} \left[ \left( \mu + \sigma_* \mu_T \right) \frac{\partial k}{\partial x_j} \right]. \tag{2.61}
\]

Specific dissipation rate equation (the $\omega$-equation):

\[
\frac{\partial \omega}{\partial t} + \rho U_j \frac{\partial \omega}{\partial x_j} = \frac{\omega}{k} \sigma_{ij} \frac{\partial U_i}{\partial x_j} - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \mu + \sigma \mu_T \right) \frac{\partial \omega}{\partial x_j} \right]. \tag{2.62}
\]

Turbulent viscosity is modeled (on dimensional grounds) as

\[
\mu_T = \frac{k}{\omega}. \tag{2.63}
\]

**Model constants** (closure coefficients)

\[
\alpha = \frac{5}{9}, \quad \beta = \frac{3}{40}, \quad \beta^* = \frac{9}{100}, \quad \sigma_\omega = \frac{1}{2}, \quad \sigma_* = \frac{1}{2}. \tag{2.64}
\]

As for the $k - \epsilon$ model, we have six equations for six unknowns ($U_1$, $U_2$, $U_3$, $P$, $k$, $\omega$) so that the system is closed.
Chapter 3

Turbulent Boundary Layers

An order-of-magnitude estimation for turbulent shear layer equations is performed and simplified equations are obtained. Then, the simplified equations are reviewed for the case of turbulent boundary layer, i.e. when the shear flow is bounded by a wall. Turbulent boundary layers are divided into inner layer (or wall layer) and outer layer (or defect layer). Convective transport terms are important only in the outer layer whereas the viscous term becomes important in the wall layer. In between, there is an overlap layer where both the convective transport and the viscous term are negligible. This is known as the logarithmic overlap region. It can be thought to mutually belong to the inner layer and to the outer layer.

3.1 The Turbulent Shear Layer Equations

For the moment we will restrict ourselves to free shear flows (i.e. flow with velocity gradient in the transverse direction that develop in the absence of boundaries, see Figure 3.1) As we shall see below, the basic approximations can be applied to all "thin" (or slowly growing) shear flows with or without a surface.

![Figure 3.1: Shear layer.](image)

We will consider only flows which are two-dimensional (although similar considerations
can be applied to flows that are axisymmetric in the mean). In effect, this is exactly the same as assuming the flow is homogeneous in the third direction. Also we shall restrict our attention to flows which are statistically stationary, so that time derivatives of time-averaged quantities can be neglected. Moreover, we confine our attention to Newtonian flows of constant density.

Since we are working in two dimensions where the flow exhibits totally different properties in the \( x \) and \( y \) directions, it will be easier to abandon tensor notation for the moment, and use the symbols \( x, U, u \) for the streamwise direction, mean and fluctuating velocities respectively, and \( y, V, v \) for the cross-stream. Also, the system may be curvilinear, but we assume that the radius of curvature of the body surface is small. Figure 3.1 shows a boundary layer represented schematically in the chosen coordinate system. We start from the stationary mean momentum equations (see Equation 2.21 from Chapter 2) written using the new notation:

**x-component**

\[
U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial x} - \frac{\partial \langle u^2 \rangle}{\partial x} - \frac{\partial \langle uv \rangle}{\partial y} + \nu \frac{\partial^2 U}{\partial x^2} + \nu \frac{\partial^2 U}{\partial y^2}.
\]

**y-component**

\[
U \frac{\partial V}{\partial x} + V \frac{\partial V}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial y} - \frac{\partial \langle v^2 \rangle}{\partial y} - \frac{\partial \langle uv \rangle}{\partial x} + \nu \frac{\partial^2 V}{\partial x^2} + \nu \frac{\partial^2 V}{\partial y^2}.
\]

In addition, we have the two-dimensional mean continuity equation (see Equation 2.16 from Chapter 2) which reduces to:

\[
\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} = 0.
\]

### 3.1.1 Order of Magnitude Estimates

In order to obtain the reduced equations governing the flow in a turbulent shear layer, we will make an order of magnitude estimate for each of the terms. We do not expect changes of quantities in the \( x \)-direction to scale the same as changes in the \( y \)-direction (this being the whole idea of thin shear flow). We will pick a length scale, \( L \), characteristic of changes or mean quantities in the \( x \)-direction, i.e.

\[
\frac{\partial}{\partial x} \sim \frac{1}{L},
\]

where \( \sim \) means "of the order of magnitude of”. We do the same for changes of mean quantities in the \( y \)-direction by defining a second length scale, \( \delta \). Therefore

\[
\frac{\partial}{\partial y} \sim \frac{1}{\delta}.
\]
3.1 The Turbulent Shear Layer Equations

the difference between the velocities at the ends of the shear layer (see Figure 3.1). From the considerations above we have

\[ \frac{\partial U}{\partial x} \sim \frac{\Delta U_s}{L}. \]  (3.6)

Now we estimate the velocity scale for \( V \), the cross-stream mean velocity component. From the continuity equation (3.3) we know that:

\[ \frac{\partial V}{\partial y} = -\frac{\partial U}{\partial x}. \]  (3.7)

If there is no mean cross flow in the free stream, the scale of \( V \) is the same as the scale for changes in \( V \). Therefore,

\[ \frac{\partial V}{\partial y} \sim \frac{V_s}{\delta}. \]  (3.8)

It follows immediately that the order of magnitude of the cross-stream velocity is

\[ V_s \sim \frac{\Delta U_s \delta}{L}. \]  (3.9)

The mean pressure gradient terms are difficult to estimate beforehand. Therefore it is better to simply leave this term, and see what is left at the end. In the estimates below we will use a question mark for the pressure gradient term.

Now we have estimated the order of magnitude of all the terms except the turbulence terms. For most problems this turns out to be pretty straightforward. The energy is distributed very evenly so that the three components of fluctuating velocity are usually about the same order of magnitude. If we choose the turbulence scale as simply \( u \), then \( \langle u^2 \rangle \sim u^2 \), \( \langle v^2 \rangle \sim u^2 \), and \( \langle w^2 \rangle \sim u^2 \). We look now at the Reynolds shear stress components like \( \langle uv \rangle \). Figure 3.2 shows a plot of \( v \) versus \( u \). As can be seen from the plot, these values tend to be well-correlated. The maximum value of \( \langle uv \rangle / u_{rms} v_{rms} \) is about 0.4, where \( u_{rms} = \sqrt{\langle u^2 \rangle} \) and \( v_{rms} = \sqrt{\langle v^2 \rangle} \).

Thus, our choice for the order of magnitude is: \( \langle uv \rangle \sim u^2 \).

3.1.2 The Streamwise Momentum Equation

We look now at the \( x \)-component of the mean momentum equation and write below each term its order of magnitude:

\[ \begin{align*}
U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} = & -\frac{1}{\rho} \frac{\partial P}{\partial x} - \frac{\partial \langle u^2 \rangle}{\partial x} - \frac{\partial \langle uv \rangle}{\partial y} + \nu \frac{\partial^2 U}{\partial x^2} + \nu \frac{\partial^2 U}{\partial y^2}.
\end{align*} \]

\( \begin{align*}
\frac{\Delta U_s}{L} + \frac{\Delta U_s^2}{L} = & ? \frac{u^2}{L} \frac{u^2}{\delta} \nu \frac{\Delta U_s}{L^2} \nu \frac{\Delta U_s}{\delta^2}
\end{align*} \)

To have a flow at all, at least one of the terms on the LHS has to survive. Since the first term is larger, we rescale all the other terms by dividing all the estimates by \( U_s \Delta U_s / L \). Now we have:
3.1 The Turbulent Shear Layer Equations

Figure 3.2: An example of \( v \) versus \( u \) (so called quadrant plot) in the wall layer at \( y^+ = 19 \). Statistical bias towards the 2nd and 4th quadrants is clearly seen. Reproduced from [11].

\[
U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} = - \frac{1}{\rho} \frac{\partial P}{\partial x} - \frac{\partial (u^2)}{\partial x} - \frac{\partial (uv)}{\partial y} + \nu \frac{\partial^2 U}{\partial x^2} + \nu \frac{\partial^2 U}{\partial y^2}.
\]

We are considering turbulent shear flow. Therefore we are interested in flows at high Reynolds number. This means the viscous terms in the mean momentum equation are negligible, which in turn means that:

\[
\frac{U_s L}{\nu} \gg 1,
\]

and

\[
\frac{U_s \delta}{\nu} \gg \frac{L}{\delta}.
\]

The second criterion is much more stringent, since \( L/\delta \) is typically of order 10. When \( U_s \delta/\nu > 1000 \), the contributions of the viscous stresses are negligible, at least as far as the \( x \)-component of the mean momentum equation is concerned.

Since we are considering the turbulent boundary layer (which would include turbulent terms), we cannot throw away all the turbulence terms on the RHS of the equation. The
biggest turbulence term is the one involving the Reynolds shear stress, \( \partial < uv > / \partial y \), which we have estimated to be of the order of \( [u^2/(U_s \Delta U_s)](L/\delta) \). Hence, we will keep at least one turbulence term only if

\[
\frac{\delta}{L} \sim \frac{u^2}{U_s \Delta U_s}.
\] (3.12)

### 3.1.3 The Transverse Momentum Equation

Now we must also consider the transverse or \( y \)-momentum equation. It cannot be considered independently from the \( x \)-momentum equation. We would never scale a vector like \((a, b, c)\) by multiplying only one component by \( S \) to produce \((Sa, b, c)\), since this would change the direction of the vector. Similarly, we must not do this for a vector equation (when doing order of magnitude estimates). We have already decided that the first term on the LHS of the \( x \)-momentum equation was the term we had to keep, and we divided by its order of magnitude, \( U_s \Delta U_s / L \), to make sure it was of order one. Thus we have already decided how we are going to scale all the components of the vector equation, and so we must do exactly the same thing here. But first we must estimate the order of magnitude of each term, exactly as before. Using our previous results, we get:

\[
U \frac{\partial V}{\partial x} + V \frac{\partial V}{\partial y} = - \frac{1}{\rho} \frac{\partial P}{\partial y} - \frac{\partial < uv >}{\partial x} - \frac{\partial < v^2 >}{\partial y} + \nu \frac{\partial^2 V}{\partial x^2} + \nu \frac{\partial^2 V}{\partial y^2}.
\]

Now dividing each term by \( U_s \Delta U_s / L \), exactly as before, we obtain:

\[
\frac{U}{L} \frac{\partial V}{\partial x} + \frac{V}{L} \frac{\partial V}{\partial y} = - \frac{1}{\rho} \frac{\partial P}{\partial y} - \frac{\partial < uv >}{\partial x} - \frac{\partial < v^2 >}{\partial y} + \frac{\nu}{U_s L} \frac{\partial^2 V}{\partial x^2} + \frac{\nu}{U_s L} \frac{\partial^2 V}{\partial y^2}.
\]

None of the mean convection terms are of order one. In fact, the only estimated term that is of order one in the whole equation is \( \partial < v^2 > / \partial y \), and only because we have already agreed that \( (u^2 / U_v \Delta U_s)(L/\delta) \) had to be of order one to keep a turbulence term in the \( x \)-momentum equation. As we do not want an equation with only a single term equal to zero, we use the pressure term to balance it, so to first order in \( \delta / L \sim u^2 / U_s \Delta U_s \), the \( y \)-momentum equation simply reduces to:

\[
0 = - \frac{1}{\rho} \frac{\partial P}{\partial y} - \frac{\partial < v^2 >}{\partial y}.
\] (3.13)

The interpretation of this equation is simple. It says that the change of the mean pressure in the transverse direction is only due to the transverse gradient of the transverse component of the Reynolds normal stress, \( < v^2 > \). We can integrate Equation 3.13 across the shear layer from a given value of \( y \) to a point in the free stream (represented as \( P(x, \infty) = P_\infty \)) to obtain:

\[
P(x, y) = P(x, \infty) - \rho < v^2 >.
\] (3.14)

This equation can be substituted into the \( x \)-momentum equation and obtain a single momentum equation, as will be shown below.
### 3.2 Wall-Bounded Turbulent Flow

Starting from this section we go one step further and consider the flow in a turbulent boundary layer, which is, basically, a wall-bounded shear flow. The schematic representation is given in Figure 3.3

![Figure 3.3: Flat-plate boundary layer.](image)

The presence of a wall introduces new effects on the flow. They arise from the wall boundary conditions. In particular:

- **The kinematic boundary condition** demands that the normal velocity of the fluid on the surface be equal to the normal velocity of the surface. This means there can be no-flow through the surface. Since the velocity normal to the surface cannot just suddenly vanish, the kinematic boundary condition ensures that the normal (transverse) velocity components in wall-bounded flows are usually much less than in free shear flows.

- **The no-slip boundary condition** demands that the velocity component tangential to the wall is the same as the tangential velocity of the wall. If the wall is at rest, then the no-slip condition demands the tangential flow velocity to be identically zero at the surface.

As we have seen from the analysis in the previous section, molecular viscosity becomes negligible at high Reynolds numbers. But in order to satisfy the no-slip condition at the wall, we need to retain at least one viscous stress term. As it will be shown below, this implies that there must be at least two length scales in the flow (this idea was originally proposed by Prandtl\(^1\) in his famous theory for wall boundary layers [3]), unlike the free shear flows we considered in the previous section for which the mean flow could be characterized by only a single length scale. The second length scale characterizes changes normal to the wall, and is chosen in such a way that at least one viscous term survives.

In the next subsections the equation for momentum in the turbulent boundary layer is derived. As it will be shown, there is a need for another "boundary layer" inside this boundary layer in order to satisfy the no-slip boundary condition. Therefore the terminology of "inner" and "outer" boundary layers is used.

---

\(^1\)Prandtl was a famous German professor at Göttingen in the early 19th century, and founder of the famous institute there which dominated much of 20th century fluid dynamics thinking, well into the second half of the century. Among his most famous students were T. Von Karman, H. Schlichting, and J. Nikuradse.
3.2 Wall-Bounded Turbulent Flow

3.2.1 The "Outer" Turbulent Boundary Layer

From the previous section we have the reduced streamwise and transverse momentum equations:

\[
\frac{U}{\partial x} + V \frac{\partial U}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial x} - \frac{\partial \langle u^2 \rangle}{\partial x} - \frac{\partial \langle uv \rangle}{\partial y}
\]

\[
P(x,y) = P_\infty - \rho < v^2 >.
\] (3.15)

We can combine these equations to obtain a single equation:

\[
\frac{U}{\partial x} + V \frac{\partial U}{\partial y} = -\frac{1}{\rho} \frac{\partial P_\infty}{\partial x} - \frac{\partial \langle uv \rangle}{\partial y} - \frac{\partial}{\partial x} \left[ \langle u^2 \rangle - \langle v^2 \rangle \right].
\] (3.17)

In the case of boundary layers, the velocity at one end of the layer is zero. Therefore, \(U_s - \Delta U_s = 0\) (see Figure 3.1) and \(\Delta U_s = U_s = U_\infty\). This means that we do not need anymore two different scales for the velocity and the change in velocity (as we needed for free shear flows). The last term is the gradient of the difference in the normal Reynolds stresses, and it is of order \(u^2/(U_s \Delta U_s) = u^2/U_s^2\) compared to the others, so is usually just ignored.

There are no viscous terms in the equation. Without viscous stresses acting near the wall to slow down the flow, the solution cannot adjust itself to zero velocity at the wall. This difficulty is addressed by the introduction of another boundary layer inside the "outer" boundary layer.

3.2.2 The "Inner" Turbulent Boundary Layer

We know that we cannot satisfy the no-slip condition unless we can figure out how to keep a viscous term in the governing equations. And we know there can be such a term only if the mean velocity near the wall changes rapidly enough so that the term \(\nu \frac{\partial^2 U}{\partial y^2}\) remains, no matter how small the viscosity becomes. In other words, we need a length scale for changes in the \(y\)-direction very near the wall which enables us to keep a viscous term in our equations. This new length scale, let’s call it \(\eta\), is going to be much smaller than \(\delta\), the turbulent boundary layer thickness. To decide how much smaller, we need to go back and look at the full equations again, and rescale them for the near wall region. To do this, we need to first decide how the mean and turbulence velocities scale near the wall. We are so close to the wall and the velocity has dropped so much (because of the no-slip condition) that it makes no sense to characterize anything by \(U_s\). But we do not have any way of knowing yet what this scale should be, so let’s just call it \(u_w\) and define it later. Also, it is known from experiments that the turbulence intensity near the wall is relatively high (30% or more). This means that there is no point in distinguishing between the turbulence scale and the mean velocity scale, we can just use \(u_w\) for both. Finally we will still use \(L\) to characterize changes in the \(x\)-direction, since these will vary no more rapidly than in the outer boundary layer. For the complete near-wall \(x\)-momentum equation we estimate:

\[
\frac{\partial U}{\partial x} + \frac{\partial U}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial x} - \frac{\partial \langle u^2 \rangle}{\partial x} - \frac{\partial \langle uv \rangle}{\partial y} + \nu \frac{\partial^2 U}{\partial x^2} + \nu \frac{\partial^2 U}{\partial y^2}.
\]

\[
\frac{u_w}{L} \frac{\partial U}{\partial x} + \frac{\partial U}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial x} - \frac{\partial \langle u^2 \rangle}{\partial x} - \frac{\partial \langle uv \rangle}{\partial y} + \nu \frac{\partial^2 U}{\partial x^2} + \nu \frac{\partial^2 U}{\partial y^2}.
\]
where we have used the continuity equation to estimate the $y$-velocity as $v_w \sim u_w \eta / L$ near the wall:

$$\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} = 0,$$

where

$$\frac{\partial U}{\partial x} \sim \frac{u_w}{L} \quad \text{and} \quad \frac{\partial V}{\partial y} \sim \frac{v_w}{\eta}.$$

Therefore

$$v_w \sim \frac{u_w \eta}{L}.$$

Now, as before, we have to decide which terms we have to keep and then compare the others with them. We insist that at least one viscous term survives. Since the largest is of order $\nu u_w / \eta^2$, we can divide by it to obtain:

$$U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} = - \frac{1}{\rho} \frac{\partial P}{\partial x} - \frac{\partial (u^2)}{\partial x} - \frac{\partial (uv)}{\partial y} + \nu \frac{\partial^2 U}{\partial x^2} + \nu \frac{\partial^2 U}{\partial y^2}.$$

Now, by our choice of $\eta$ (characteristic thickness of the "inner" layer) we can decide whether the Reynolds shear stress term survives or not in this particular "inner" boundary layer. Thus, we can choose the layer to be thick enough so that $u_w \eta / \nu \sim 1$ or thin enough such that $u_w \eta / \nu \to 0$. The most general choice is to pick $\eta \sim \nu / u_w$ so that the Reynolds shear stress remains too. (This is called the distinguished limit in asymptotic analysis.) As a consequence of this choice, the "inner" Reynolds number (computed with respect to the dimension of the "inner" layer) is unity; i.e., $Re_\eta = u_w \eta / \nu = 1$, meaning that viscous and inertial terms are about the same. This is precisely why we defined $\eta$ as we did in the first place: to ensure that viscosity was important.

When we choose $\eta \sim \nu / u_w$, then all the other terms vanish, except for the pressure gradient, the viscous term, and the Reynolds shear stress gradient. In the turbulence literature ([27], [20]), when analysing the structure of turbulent boundary layers (and deriving the law of the wall), the assumption is made that the streamwise pressure gradient is zero or at least close to zero. In [20] this is explained by the use of the Bernoulli’s principle

$$\frac{U^2}{2} + \frac{P}{\rho} = constant \quad (3.18)$$

and by taking the $x$ derivative of the equation, to give:

$$\frac{\partial P}{\partial x} = -\rho U \frac{\partial U}{\partial x}. \quad (3.19)$$

To be able to apply Bernoulli’s principle (which is valid for inviscid, non-turbulent flow) it is assumed that the external flow (free stream) is inviscid and consists of intermittent turbulent/non-turbulent motion. Thus, if the free stream is at constant mean velocity, then $\frac{\partial P}{\partial x}$ is exactly zero. Other authors are making order-of-magnitude estimates directly for the so-called ZPG (Zero-Pressure Gradient) boundary layer, so that a zero
3.2 Wall-Bounded Turbulent Flow

pressure gradient in the $x$ direction is assumed from the beginning. We have to mention here that flat-plate boundary layer was taken as a general case of flow near a boundary, so that a universal law of the wall for all bounded flows can be derived. However, if we think of pipe flow, the reasoning above is not fully applicable as there is no free stream in pipe flow.

After the analysis above, our mean momentum equation for the near wall region reduces to:

$$0 = \frac{\partial}{\partial y} \left[ - < uv > + \nu \frac{\partial U}{\partial y} \right].$$

(3.20)

We integrate Equation 3.20 from the wall to location $y$ to obtain:

$$0 = -\left\{ < uv > - < uv > \big|_{y=0} \right\} + \nu \left\{ \frac{\partial U}{\partial y} - \frac{\partial U}{\partial y} \bigg|_{y=0} \right\}.$$

(3.21)

From the kinematic and no-slip boundary conditions at the wall we immediately know that $< uv > \big|_{y=0} = 0$. We also know that the wall shear stress is given by:

$$\tau_w = \mu \frac{\partial U}{\partial y} \bigg|_{y=0}.$$

(3.22)

Substituting this we obtain our equation for the very near wall as:

$$\frac{\tau_w}{\rho} = - < uv > + \nu \frac{\partial U}{\partial y}.$$

(3.23)

This is an important result of boundary layer theory; namely that the total stress in the near-wall layer is constant, where the total stress consists of the stress due to viscous forces and the Reynolds stress (stress due to turbulence). Therefore, the wall layer is referred to as the Constant Stress Layer.

Figure 3.4 shows the normalized shear stress in a turbulent boundary layer. The solid line represents the total shear stress, the dashed line - the viscous shear stress $\frac{\nu u^2}{\nu} \frac{\partial U}{\partial y}$, and the dotted line - the turbulent shear stress $-\frac{< uv >}{u^2}$. The horizontal lines separate the sublayers of the "inner" layer. They will be introduced in the next section.

Now we need to resolve the important questions of what is $u_w$, our inner velocity scale. It is customary to define something called the friction velocity, usually denoted as $u_*$, by:

$$u_*^2 = \frac{\tau_w}{\rho}.$$

(3.24)

Now using this, Equation 3.23 can be rewritten as:

$$u_*^2 = - < uv > + \nu \frac{\partial U}{\partial y}.$$

(3.25)

It makes sense to choose $u_w = u_*$; in other words, the friction velocity is the appropriate scale velocity for the near-wall region. It follows immediately from our considerations above that the inner length scale is $\eta \sim \nu / u_*$.

Finally, we can use our new length scale to define where we are in this near wall layer. In fact, we can introduce a new dimensionless coordinate called $y^+$ and defined as:

$$y^+ = \frac{y}{\eta} = \frac{yu_*}{\nu} \Rightarrow \frac{\partial}{\partial y} = \frac{u_*}{\nu} \frac{\partial}{\partial y^+}.$$
3.3 Sublayers of the Constant Stress Layer

Figure 3.4: Normalized shear stress in a flat-plate turbulent boundary layer.

We can non-dimensionalize Equation 3.23 using inner variables and rewrite it as:

\[ 1 = \sigma^+ + \frac{\partial U^+}{\partial y^+}, \]  

(3.26)

where

\[ U^+ = \frac{U(x, y)}{u_*} \]  

(3.27)

and

\[ \sigma^+ = -\frac{\langle uv \rangle}{u_*^2}. \]  

(3.28)

By contrast an "outer" dimensionless coordinate, \( \bar{y} \), is defined by:

\[ \bar{y} = \frac{y}{\delta}. \]  

(3.29)

In terms of these coordinates the outer equations for the mean flow are generally considered valid outside of \( y^+ = 30 \). And the inner equations are needed inside of \( \bar{y} = 0.1 \) if we take \( \delta = \delta_{0.99} \), where \( \delta_{0.99} \) is defined to be equal to the value of \( y \) at which the mean velocity in the boundary layer is 99% of its free stream value, \( U_\infty \).

The ratio of \( y^+ \) to \( y \) is the local Reynolds number \( \delta^+ \), where

\[ \delta^+ = \frac{\delta u_*}{\nu}. \]  

(3.30)

The higher the value of \( \delta^+ \), the closer the inner layer will be to the wall relative to \( \delta \). On the other hand, since the wall friction (and hence \( u_* \)) drops with increasing distance downstream, both the outer boundary layer and inner boundary grow in physical variables (i.e., the value of \( y \) marking their outer edge increases).
Figure 3.5: Sketch showing the various regions of the turbulent boundary layer in inner and outer variables.
3.3 Sublayers of the Constant Stress Layer

The viscous stress and Reynolds stress cannot both be important all the way across the constant stress layer. In fact, inside \( y^+ = 3 \), the Reynolds stress term is negligible (to within a few percent), so very near the wall Equation 3.23 reduces to:

\[
v^2_s \approx \nu \frac{\partial U}{\partial y},
\]

or in inner variables

\[
1 \approx \frac{\partial U^+}{\partial y^+}.
\]

It can immediately be integrated to obtain the velocity near the wall as:

\[
U^+(y^+) = y^+ + \text{constant},
\]

and, due to the no-slip boundary condition, the constant term vanishes to give:

\[
U^+(y^+) = y^+,
\]

or in physical variables

\[
U(y) = \frac{u^2_s y}{\nu}.
\]

Because of the linear relation between \( U \) and \( y \) this subregion very near the wall is referred to as the linear sublayer (sometimes the name laminar sublayer is also used).

Sometimes the extent of the linear sublayer is considered to be up to \( y^+ = 5 \). However, by \( y^+ = 3 \) the Reynolds stress has already begun to evolve, making the approximations inexact. In other words, the linear approximation is only good to within about 10% at \( y^+ = 5 \); and the linear approximation deteriorates rapidly outside this region. This linear region in the mean velocity at the wall is one of the very few exact solutions in turbulence modeling. There are no adjustable constants (as there will be in the inertial sublayer, whose velocity profile is described by a log function with two coefficients). It presents great advantages for experimentalists. They only need to measure the velocity down to \( y^+ = 3 \) to be able to get an excellent estimate of the wall shear stress. However, few experiments can resolve this region.

The linear sublayer is part of the viscous sublayer, which extends until the mean flow is dominated by the Reynolds stress alone at about \( y^+ = 30 \).

As we move out of the linear region very close to the wall the Reynolds stress rapidly develops until it overwhelms the viscous stress. Also, as we move outward, the mean velocity gradient slowly drops until the viscous stress is negligible compared to the Reynolds shear stress. In fact, by \( y^+ = 30 \), the viscous shear stress is less than a percent or so of the Reynolds shear stress. Thus outside this point we have only:

\[
u^2_s = - \langle uv \rangle.
\]

In other words, the Reynolds shear stress is itself nearly constant. Viscous effects on the mean flow are negligible, and only the inertial terms from the fluctuating motion remain. This region is referred to as the inertial sublayer.

Figure 3.5 summarizes the important regions of a turbulent boundary layer (in external flow). The constant stress layer has two parts: a viscous sublayer and an inertial sublayer.
The viscous sublayer itself has two identifiable regions: the linear sublayer where only viscous shear stresses are important, and a buffer layer in which both viscous and Reynolds shear stresses are important. Note that in case $\delta^+$ is less than approximately 3000, then the inertial layer cannot be present.

### 3.4 The Law of the Wall

To derive the so-called law of the wall, we will use the eddy viscosity approximation introduced in Chapter 2 and apply it for the momentum equation in the inertial sublayer. From Equation 2.24 we have:

$$-\rho < uv >= \mu_T \left( \frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} \right).$$

(3.37)

Since we are in a boundary layer the changes in the $x$ direction ($\frac{\partial V}{\partial x}$) are negligible in comparison with the changes in the $y$ direction. Therefore the above equation becomes:

$$-\rho < uv >= \mu_T \frac{\partial U}{\partial y}.$$  

(3.38)

Now we plug this into our momentum equation 3.36 to obtain:

$$u^2_* = \frac{\mu_T}{\rho} \frac{\partial U}{\partial y} = \nu_T \frac{\partial U}{\partial y},$$

(3.39)

where

$$\nu_T = \frac{\mu_T}{\rho}. \tag{3.40}$$

Using Prandtl’s assumption for the turbulent viscosity [24], which says that the turbulent viscosity $\nu_T$ near the wall scales with the distance from the wall

$$\nu_T = \kappa u_* y,$$ 

(3.41)

where $\kappa$ is a constant, Equation 3.39 becomes

$$\frac{\partial U}{\partial y} = \frac{u_*}{\kappa y}.$$ 

(3.42)

It can be immediately integrated to yield the well-known Law of the Wall:

$$U = \frac{u_*}{\kappa} \ln y + \text{const.}.$$ 

(3.43)

We can obtain the dimensionless version of the law by using the dimensionless velocity and normal distance defined as in the previous section:

$$U^+ = \frac{U}{u_*},$$

(3.44)

$$y^+ = \frac{u_* y}{\nu},$$

(3.45)

and substituting them into Equation 3.42 to give
\[ \frac{\partial U^+}{\partial y^+} = \frac{1}{\kappa y^+}. \quad (3.46) \]

Again, this ODE can be easily integrated, so that

\[ U^+ = \frac{1}{\kappa} \ln y^+ + B. \quad (3.47) \]

It is referred to as the *Universal Law of the Wall*. It is called *universal* because the parameters \( \kappa \) and \( B \) do not depend on the Reynolds number (thus, they are universal constants for all flows). The coefficient \( \kappa \) is known as the *Kármán constant*, and \( B \) is a dimensionless constant. These coefficients have been found by fitting the law to a large amount of experimental data [7]. The values given in the literature are:

\[
\begin{align*}
\kappa & \approx 0.40 \ldots 0.41, \\
B & \approx 5.0 \ldots 5.5.
\end{align*}
\quad (3.48, 3.49)
\]

As it was mentioned before, the velocity profile in the linear sublayer (where the logarithmic law fails because the velocity does not vanish at \( y = 0 \)) is approximated by the relation 3.34. Figure 3.6 shows the velocity profile consisting of a region with linear increase in velocity right near the wall and a logarithmic behaviour in the overlap layer.

![Figure 3.6: Velocity profile in a turbulent boundary layer.](image)

Ignoring the region of transition in which the velocity profile gradually changes from one regime to the other, we can try to connect the two, by solving the following transcendental equation:

\[ y^+ = \frac{1}{0.40} \ln y^+ + 5.0. \quad (3.50) \]

The numerical solution is \( y^+ = 10.8 \), which falls exactly into the buffer layer, between the linear and the inertial sublayers.

It must be mentioned that the Logarithmic Law of the Wall is not undisputable. After all, Prandtl’s assumption 3.41, used in the derivation, is based only on dimensional
3.4 The Law of the Wall

grounds. The scaling in the inertial sublayer (also referred to as overlap region) of turbulent wall-bounded flows has long been the source of controversy. Barenblatt et al. [4] developed theories showing that *power laws* are more suitable for describing velocity profiles in wall-bounded turbulent flows. Until recently this controversy could not be addressed because measurements did not span a sufficient range of Reynolds number. However, in 1997 new experiments conducted by Zaragola et al. [29] have shown that at sufficiently high Reynolds numbers, the mean velocity profile in the overlap region is found to be better represented by a log law than a power law. These results suggest a theory of complete similarity instead of incomplete similarity, contradicting the theories developed by Barenblatt et al.
Chapter 4

The Law of the Wall as Boundary Condition

Because of the large velocity gradients arising in the region near the wall, it requires special treatment. Moreover, the flow in the near-wall region is no longer turbulent (at least not everywhere) so that the assumptions made while deriving the turbulence models are not valid. Two approaches for resolving the near-wall region are presented. Then, one of them, the law-of-the-wall approach is described and details of its use as boundary condition for computational purposes are given.

4.1 Computational Approaches for the Near-Wall Region

Traditionally, there are two approaches to modeling the near-wall region. In one approach, the turbulence models are modified to enable the viscosity-affected region to be resolved with a mesh all the way to the wall, including the viscous sublayer. In another approach, the viscosity-affected inner region (viscous sublayer and buffer layer) is not resolved. Instead, the semi-empirical wall functions (discussed in the previous section) are used to bridge the viscosity-affected region between the wall and the fully-turbulent region. The use of wall functions obviates the need to modify the turbulence models to account for the presence of the wall. These two approaches are depicted schematically in Figure 4.1.

In most high-Reynolds-number flows, the wall function approach substantially saves computational resources, because the viscosity-affected near-wall region, in which the solution variables change most rapidly, does not need to be resolved. The wall function approach is popular because it is economical, robust, and reasonably accurate. It is a practical option for the near-wall treatment in industrial flow simulations.

4.2 Boundary Conditions

The boundary conditions derived from the wall functions (law of the wall) are applied at a location $y = y_p$ in the log-law region ($y_p$ is chosen such that $y_p^+$ is between 30 and 100). We use the subscript 'p' to indicate quantities evaluated at $y_p$, such as $U_p$, $k_p$, $\epsilon_p$, $\nu T_p$.

The law of the wall (Equation 3.43) introduces an additional unknown, $u_*$, which is clearly not very convenient. Therefore, we try to express $u_*$ in terms of already existing variables. Next, we will prove that the following relation holds:

$$u_* = C_\mu^{1/4} k_p^{1/2}.$$
We start with Equation 3.39 evaluated at \( y_p \)

\[
\frac{u_*^2}{\nu} = \nu_T \frac{\partial U_p}{\partial y}.
\]

From Equations 3.40 and 2.58

\[
\nu_T = C_\mu \frac{k_p^2}{\epsilon_p},
\]

then

\[
\frac{u_*^2}{\nu} = C_\mu \frac{k_p^2}{\epsilon_p} \frac{\partial U_p}{\partial y},
\]

Now we use the law of the wall (Equation 3.43) to compute the velocity derivative

\[
\frac{\partial U_p}{\partial y} = \frac{u_*}{\kappa y_p},
\]

and plug it in Equation 4.2

\[
\frac{u_*^2}{\nu} = C_\mu \frac{k_p^2}{\epsilon_p} \frac{u_*}{\kappa y_p},
\]

\[
\Rightarrow \epsilon_p = \frac{C_\mu k_p^2}{u_* \kappa y_p}.
\]

Thus, we obtained an equation relating the three unknowns: \( u_* \), \( k_p \) and \( \epsilon_p \). Next, a second equation for the three unknowns will be obtained. This will allow us to express \( u_* \) only in terms of one unknown, \( k_p \).
We consider the turbulent flow to be in an equilibrium state, which means that the production of turbulent kinetic energy $P_k$ must balance the dissipation of turbulent kinetic energy $\rho \epsilon$:

$$\rho \epsilon = P_k. \quad (4.6)$$

From its definition (see Equation 2.47 in Chapter 2)

$$P_k = -\rho <u_i u_j> \frac{\partial U_j}{\partial x_i}. \quad (4.7)$$

If we use the eddy-viscosity hypothesis (see Section 2.3, Equation 2.24) to estimate the Reynolds stress, it can be written (in 2D):

$$P_k = -\rho <u_i u_j> \frac{\partial U_j}{\partial x_i} = \mu_T \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j}. \quad (4.8)$$

We replace the tensor notation with the usual one:

$$P_k = \mu_T \left( \frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} \right) \frac{\partial U}{\partial y}. \quad (4.9)$$

We are considering boundary layer flow, so there is no flow in the cross-stream direction ($V = 0$). Hence

$$\frac{\partial V}{\partial x} = 0,$$

and

$$P_k = \mu_T \left( \frac{\partial U}{\partial y} \right)^2. \quad (4.10)$$

By combining the Equations 4.6 and 4.10, and evaluating all the unknown quantities at $y_p$, we have:

$$\epsilon_p = \frac{\mu_T}{\rho} \left( \frac{\partial U_p}{\partial y} \right)^2 = \nu_T \left( \frac{\partial U_p}{\partial y} \right)^2. \quad (4.11)$$

We use Equation 4.1 for the kinematic viscosity and Equation 4.3 for the cross-stream derivative of $U_p$, to get:

$$\epsilon_p = C_\mu \frac{k_p^2}{\epsilon_p} \frac{u_\tau^2}{\kappa y_p^2}, \quad (4.12)$$

$$\Rightarrow \epsilon_p = C_\mu^{1/2} \frac{k_p u_\tau}{\kappa y_p}. \quad (4.13)$$

Now, the right-hand sides of Equations 4.5 and 4.13 must be equal:

$$\frac{C_\mu k_p^2}{u_* \kappa y_p} = \frac{C_\mu^{1/2} k_p u_\tau}{\kappa y_p}, \quad (4.14)$$

from which we arrive at the expression for $u_*$:

$$u_* = C_\mu^{1/4} k_p^{1/2}. \quad (4.15)$$
4.2 Boundary Conditions

This relation is used in most of the software packages that support simulation of turbulent flow with two-equation models. However, the way it is used is different from package to package. In the following, the way the boundary conditions based on the law of the wall are implemented in COMSOL Multiphysics\textsuperscript{TM} [15], [16], is described.

We start from the law of the wall (Equation 3.47) evaluated at \( y_p \) (or, equivalently, at \( y_p^+ \)):

\[
U_p^+ = \frac{1}{\kappa} \ln y_p^+ + B.
\]

Use Equation 3.44 to replace the dimensionless velocity with the physical velocity:

\[
\frac{U_p}{u_*} = \frac{1}{\kappa} \ln y_p^+ + B.
\] (4.16)

Multiply by \( u_*^2 \)

\[
U_p u_* = u_*^2 \left( \frac{1}{\kappa} \ln y_p^+ + B \right).
\] (4.17)

Use Equation 3.24 to replace \( u_*^2 \) by \( \tau_w / \rho \), and Equation 4.15 to replace \( u_* \) by \( C_{\mu}^{1/4} k_p^{1/2} \)

\[
U_p C_{\mu}^{1/4} k_p^{1/2} = \frac{\tau_w}{\rho} \left( \frac{1}{\kappa} \ln y_p^+ + B \right).
\] (4.18)

From the last relation we can obtain an expression for the skin friction force at the wall (or wall stress), \( \tau_w \):

\[
\tau_w = \frac{\rho C_{\mu}^{1/4} k_p^{1/2}}{U_p}.
\] (4.19)

But the stress at the wall (or anywhere else) can be computed as a function of the velocity gradient (Newtonian Fluid approximation). To do this we use Equations 2.5 and 2.6 from Chapter 2, given below for convenience:

\[
\hat{T}_{ij}^{(v)} = 2 \mu_{tot} [\tilde{s}_{ij} - \frac{1}{3} \tilde{s}_{kk} \delta_{ij}],
\] (4.20)

where

\[
\tilde{s}_{ij} = \frac{1}{2} \left[ \frac{\partial \hat{u}_i}{\partial x_j} + \frac{\partial \hat{u}_j}{\partial x_i} \right].
\] (4.21)

Keeping in mind that our fluid flow is incompressible (so \( \tilde{s}_{kk} = 0 \)) and that we write the equations for the mean flow (so \( \mu_{tot} = \mu + \mu_T \)), we have:

\[
T_{ij} = (\mu + \mu_T) \left[ \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right],
\] (4.22)

or written in vectorial notation

\[
\mathbf{T} = (\mu + \mu_T) \left[ \nabla \mathbf{U} + (\nabla \mathbf{U})^T \right].
\] (4.23)

Here, \( \mathbf{T} \) is a second-order tensor. For simplicity, we did all the reasoning in this chapter by assuming that the wall coincides with the \( x \)-axis and the wall stress \( \tau_w \) turned out to be a scalar. However \( \tau_w \) is actually a vector, and we can compute it by making a projection of \( \mathbf{T} \) on the normal to the surface, \( \mathbf{n} \):
4.3 Thickness of the Wall Region

\[ \tau_w = T \cdot n = (\mu + \mu_T) [\nabla U_p + (\nabla U_p)^T] \cdot n. \] (4.24)

From Equations 4.19 and 4.24,

\[ (\mu + \mu_T) [\nabla U_p + (\nabla U_p)^T] \cdot n = \frac{\rho C_{\mu}^{1/4} k_p^{1/2}}{\kappa \ln y_p^+} U_p, \] (4.25)

where the velocity \( U_p \) on the RHS has also become a vector to account for the general case when the wall does not coincide with one of the coordinate axes. The value of \( y_p^+ \) is computed as:

\[ y_p^+ = \frac{u_\ast y_p}{\nu} \] (4.26)

The equation above is the boundary condition for \( U \), as implemented in COMSOL Multiphysics\textsuperscript{TM}. The only thing left is to find the boundary conditions for \( k \) and \( \epsilon \). In the case of turbulent kinetic energy \( k \), the answer is obvious; there should not be any transfer of energy through the wall, or, in other words, the zero-flux boundary condition:

\[ n \cdot \nabla k_p = 0. \] (4.27)

To arrive at the boundary condition for \( \epsilon \) we use the Equations 4.13 and 4.15 to obtain:

\[ \epsilon_p = \frac{C_\mu^{3/4} k_p^{3/2}}{\kappa y_p^+ \nu}. \] (4.28)

4.3 Thickness of the Wall Region

The thickness of the wall region, \( y_p \) (see Equation 4.28), is a free parameter and is prescribed before the simulation. It must be given in such a way that \( y_p^+ \) falls within 30 and 100 (the range of validity of the law of the wall). In case we want to prescribe \( y_p^+ \) directly, Equation 4.26 is used to transform the boundary condition for \( \epsilon \) to the following form:

\[ \epsilon_p = \frac{C_\mu^{3/4} k_p^{3/2} u_\ast}{\kappa y_p^+ \nu}. \] (4.29)

Now, using Equation 4.15 we arrive at the final form of the boundary condition for \( \epsilon \):

\[ \epsilon_p = \frac{C_\mu k_p^2}{\kappa y_p^+ \nu}. \] (4.30)

If the law of the wall would describe the velocity profile exactly, then, theoretically, the choice of \( y_p^+ \) (in the range between 30 and 100) would not influence the solution. However, the law of the wall is an approximate and a semi-empirical relation. Therefore, the solution does depend on the thickness of the near-wall region. We hope though, that this dependence is not too strong (again, for \( y_p^+ \) in the range between 30 and 100).

Simulations were performed to observe the influence of thickness of the wall region on the solution. The value \( y_p^+ \) spans from 6.25 up to 3200 in a geometric progression. To measure the difference between solutions a certain norm could be used. However, since we are also interested in friction factor estimation, we will use this as an indicator. Figure 4.2 shows the dependence of the friction factor on the thickness of the wall region at Reynolds number of order \( 10^6 \).
Figure 4.2: Dependance of the friction factor on the thickness of the wall region at Reynolds number of order $10^6$. Solid line - computed values, dotted line - experimental values.

Although some preliminary simulation have shown a strong dependence of $f$ on $y_p^+$ (Appendix B), after individually adjusting the mesh (by using the adaptive solver) for each value of $y_p^+$, much better results have been obtained. As can be seen from the plot, the friction factor is almost constant in the range of $y_p^+$ between 50 and 300 (between the vertical lines in the plot). In this range, the variation of $f$ is less than 1.5%. Thus, the range of validity of the law of the wall proves to be in practice from 50 to 300 (instead of 30 to 100).
Chapter 5

Turbulent Flow in Conventional Pipes

We begin with a brief review on turbulent flow in pipes. Then a simulation of a fully developed turbulent flow in a non-corrugated pipe is set up. The computational domain, boundary conditions and grid generation are described. For the simulations, a pipe with smooth walls is considered first, after which modifications to the law of the wall are made to model the effects of roughness. The simulations described in this chapter serve the purpose of validating the chosen turbulence models, constants and (especially) boundary conditions, so that they can be applied (with certain adjustments) to the more complicated case of fully developed turbulent flow in periodically corrugated pipes.

5.1 Pipe Flow

Pipes are closely associated with fluid flow. We encounter pipe flows almost everywhere - for water and gas supply at houses and offices, in fluid machinery, for transporting oil over long distances and at many other places.

A flow can be an Internal Flow or an External Flow. From the computational point of view, a flow is considered internal if boundary conditions are only imposed at the outer boundaries of the computational domain (for instance, flow inside a vessel or pipe). On the other hand, a flow is external if boundary conditions are also imposed at some inner boundary of the computational domain. Internal flows are further classified into Pipe Flows and Open Channel Flows. A pipe flow is one where the fluid fills the conduit completely (Fig. 5.1, left). The main driving force is pressure although gravity may also be contributing to the flow. On the other hand, an open channel flow is one where the fluid does not fill the conduit completely (Fig. 5.1, right). The driving force in this is gravity.

Let us consider a fluid entering a pipe. We assume that the entering flow is uniform (constant velocity). Thus, the velocity gradient in the direction perpendicular to the flow vanishes, so viscosity effects are unimportant (inviscid flow). However, as soon as the flow "hits" the pipe wall many changes take place. The most important of these is that the "no slip" condition at the wall comes into effect and viscosity imposes itself on the flow. Consequently the velocity components are each zero on the wall, i.e., $\tilde{u} = \tilde{v} = 0$. The flow adjacent to the wall decelerates continuously. We have a layer close to the wall where the velocity builds up slowly from zero at the wall to a uniform velocity towards the center of the pipe. This is the boundary layer examined in Chapter 3. Viscous effects are dominant
within the boundary layer. Outside of this layer is the inviscid core where viscous effects are negligible or absent (see Fig. 5.2).

The boundary layer is changing with the flow. It grows, meaning that its thickness increases as we move downstream. From Fig. 5.2 it is seen that the boundary layer from the walls grows to such an extent that they all merge at the centerline of the pipe. Once this takes place, the inviscid core dissappear and the flow is all viscous (here and in the rest of the discussion by viscosity we mean both molecular and turbulent viscosity) The flow is now called a fully developed flow. Once the flow is fully developed the velocity profile does not vary in the flow direction. In fact in this region the pressure gradient and the shear stress at the wall are in balance. The length of the pipe between the start and the point where the fully developed flow begins is called the entrance length. Denoted by $L_e$, the entrance length is a function of the Reynolds Number of the flow. In general,

$$\frac{L_e}{D} \approx 0.06 Re,$$  \hspace{1cm} (5.1)

$$\frac{L_e}{D} \approx 4.4 Re^{1/6},$$  \hspace{1cm} (5.2)

where $Re$ is the Reynolds Number (based on the pipe diameter).
\[ Re = \frac{vD}{\nu}. \]  

At the critical point, i.e., \( Re = 2300 \), the \( L_e/D \) value laminar flow is 138. Under turbulent conditions it ranges from 18 (at \( Re = 4000 \)) to 95 (at \( Re = 10^8 \)).

As a rule of thumb, usually we need to have a pipe with a length of 100 times its diameter to obtain a fully developed flow.

The forces acting upon the pipe flow are inertial force (spatial acceleration/deceleration due to the changing velocity profile), viscous force due to shear, and the force caused by the pressure gradient. If the pipe is horizontal, we can ignore the effects of gravity. When the flow is fully developed the pressure gradient and shear forces balance each other and the flow continues with a constant velocity profile. The pressure gradient remains constant, which implies a linear decrease of pressure in the downstream direction.

In the entrance region the fluid is decelerated near the walls and accelerated in the middle. The shear forces are responsible for the deceleration and a higher pressure gradient is needed to cause the acceleration. Thus, a balance is achieved between inertia, pressure and shear forces. The pressure gradient is not constant in this part of the flow and in fact, it decreases as shown in Figure 5.3.

Figure 5.3: Pressure distribution along the flow in a pipe.
5.2 Validation Tests for Pipe Flow and the Moody Diagram

Validation can be defined as the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model.

It is not possible to validate an entire CFD code. One can only validate the code for a specific range of applications for which there is experimental data. Thus one validates a model or simulation. Applying the code to flows beyond the region of validity is called prediction.

Validation examines if the conceptual models, computational models as implemented into the CFD code, and computational simulation agree with real world observations. The strategy is to identify and quantify error and uncertainty through comparison of simulation results with experimental data. The experiment data sets themselves might contain bias errors and random errors which must be taken into account. The accuracy required in the validation activities is dependent on the application, and so, the validation should be flexible to allow for various levels of accuracy.

The approach to Validation Assessment is to perform a systematic comparison of CFD simulation results to experimental data from a set of increasingly complex cases.

For pipe flow, one can use the so-called **Moody Diagram** as a large database of experimental data (confirmed by the "theoretical" Colebrook-White equation). The Moody diagram\(^1\) represents graphically the friction factor as a function of Reynolds number. At low Reynolds numbers, it shows the \(64/Re\) relationship for laminar flows. For higher Reynolds numbers, the diagram also incorporates a factor for the relative roughness of the walls (see Figure 5.4).

The relative wall roughness is usually written as \(e/D\) where \(e\) is the size of irregularities of the wall and \(D\) is, as before, the pipe diameter.

The Darcy-Weisbach friction factor is defined as:

\[
f = \frac{\Delta P D}{0.5 \rho U_{\text{avg}}^2 L},
\]

where \(\Delta P/L\) is the pressure gradient needed to drive the flow at average velocity \(U_{\text{avg}}\).

The wall shear stress normalized by dynamic pressure is called the skin friction coefficient.

\[
C_f = \frac{\tau_w}{0.5 \rho U_{\text{avg}}^2},
\]

where \(\tau_w\) is the shear stress at the wall. The skin friction coefficient is very closely related to the friction factor. Equilibrium of forces requires that \(\Delta P = 4LD\tau_w/D\), so that

\[
f = 4C_f.
\]

The Moody diagram reproduces the values of the famous Colebrook-White formula:

\[
\frac{1}{\sqrt{f}} = -2 \log \left( \frac{e/D}{3.7} + \frac{2.51}{Re \sqrt{f}} \right).
\]

\(^1\)The Moody diagram is based on the pioneering work of Thomas Stanton (1865-1931) who with J. R. Pannel conducted experiments on a number of pipes of various diameters, materials and fluids. At the time, the plot of this data was therefore sometimes known as the Stanton diagram. A Georgian engineer based in Germany, Johann Nikuradse (1894) extended the results by doing experiments on artificially roughened circular pipes. An American engineer Lewis F. Moody (1880-1953) prepared the diagram, in the form it is used today, for use with ordinary commercial pipes.
However, this relation is difficult to use because of its implicit nature ($f$ on both sides). Therefore, alternative explicit approximations have been sought. One of them was suggested by Haaland [10]

$$\frac{1}{\sqrt{f}} = -1.8 \log \left( \frac{e/D}{3.7} \right)^{1.11} + \frac{6.9}{Re},$$

(5.8)

which varies less than 2% from the Colebrook formula.

### 5.3 Simulation Setup

One of the objectives of this study is to evaluate the performance of two-equation models ($k-\epsilon$ and $k-\omega$) as implemented in the Finite Element Package COMSOL Multiphysics™. To do this, we will simulate the turbulent flow in a pipe and assess the validity of the results by comparing the friction factor computed from the simulation to the one given by the Moody diagram. Unless otherwise stated, a pipe with smooth walls is considered.

#### 5.3.1 Computational Domain Geometry

The fully developed and time-averaged turbulent flow in a smooth pipe is one-dimensional and axisymmetric in its nature, the only dimension being taken in the radial direction.

---

2COMSOL Multiphysics (formerly FEMLAB) is a finite element analysis and solver software package for various physics and engineering applications, especially coupled phenomena, or multiphysics. Its development was started by graduate students to Germund Dahlquist based on codes developed for a graduate course at the Royal Institute of Technology (KTH) in Stockholm, Sweden.
In other words, the flow only depends on the distance from the pipe’s centerline. In our simulation, a 2D axisymmetric model with periodic boundary conditions at inflow and outflow will be used. This is done because of two reasons:

- It gives the possibility of modifying the shape of the wall at a later time.
- The 1D axisymmetric mode is not available in COMSOL Multiphysics™ for the two-equations turbulence models.

Therefore the computational domain will consist of a short segment of the pipe, bounded by the wall at the right and a symmetry axis at the left, as shown in Figure 5.5. Since we are investigating a fully developed flow, we are not interested in any temporal behaviour, so the steady Navier-Stokes equations will be used. To include the effects of turbulence, two additional equations (Equations 2.56 and 2.57 for the $k - \epsilon$ model, and Equations 2.61 and 2.62 for the $k - \omega$ model) are added to the model, as discussed in Chapter 2.

### 5.3.2 Boundary Conditions

As can be seen from Figure 5.5, our computational domain has four distinct boundaries for which boundary conditions have to be prescribed. As explained above, we use a 2D axisymmetric model. Thus, at one of the boundaries parallel to the flow the Axial Symmetry boundary condition is prescribed. This boundary coincides with the centerline of the pipe. The other boundary parallel to the flow coincides with the wall of the pipe. Along this boundary, the law of the wall is used to prescribe the velocity at a certain distance from the wall. The use of the law of the wall as a boundary condition has been fully described in Chapter 4. Here, we only list the equations.

For the $k - \omega$ model:

\[ \mathbf{n} \cdot \mathbf{U}_p = 0, \quad (\mu + \mu_T) \left[ \nabla \mathbf{U}_p + (\nabla \mathbf{U}_p)^T \right] \cdot \mathbf{n} = \frac{\rho C_{\mu}^{1/4} k_p^{1/2}}{1/\kappa \ln y_p^+ + B} \mathbf{U}_p, \]

\[ \mathbf{n} \cdot \nabla k_p = 0, \]

\[ \omega_p = \frac{\rho k_p}{\kappa y_p^+ \mu}. \]
For the $k - \epsilon$ model:

$$n \cdot U_p = 0,$$

$$\left( \mu + \mu_T \right) \left[ \nabla U_p + (\nabla U_p)^T \right] \cdot n = \frac{\rho C_\mu^{1/4} k_p^{1/2}}{k y_p^+ \mu} U_p,$$

$$n \cdot \nabla k_p = 0,$$

$$\epsilon_p = \frac{\rho C_\mu k_p^2}{k y_p^+ \mu}.$$

In the above equations $n$ is the unit vector normal to the wall and $y_p^+$ represents the thickness of the log layer and its value is set to 100.

Now we look at the boundaries normal (perpendicular) to the flow. Certainly, they represent inflow and outflow boundaries. To use the inflow/outflow boundary condition, we need to prescribe the velocity profiles at both ends. But this is a problem here. We are interested in fully developed flow and do not know the velocity profiles of the resulting flow. One way to solve this, is to use a very long pipe with a random velocity profile at the entrance (e.g. uniform velocity profile). As it was explained in Section 5.1, to obtain fully developed flow we will need a pipe which is about 100 times longer than its diameter. This is, obviously, very expensive from the computational point of view. Moreover, because of the huge computational domain, only coarse meshes can be used, which drastically decreases the quality of the solution.

A very effective method of overcoming the above difficulties is the use of periodic boundary conditions. Their use is explained by the fact that the fully developed flow has a constant velocity profile, which means that:

$$U(r, 0) = U(r, L),$$

$$V(r, 0) = V(r, L).$$

where $L$ is the length of the computational in the axial direction (set as $L = 0.02$).

Also, the turbulence-related quantities ($k$, and $\epsilon$ or $\omega$) have the same values at inflow and outflow:

$$k(r, 0) = k(r, L),$$

$$\epsilon(r, 0) = \epsilon(r, L),$$

$$\omega(r, 0) = \omega(r, L).$$

The use of periodic boundary conditions brings a huge decrease of computational costs and facilitates the use of finer meshes. Later, pipes with (periodic) corrugations can be simulated by using the same periodic boundary conditions. In this case, at least one period has to be included in the computational domain.

The last issue to be addressed is the boundary condition for pressure at the inflow and outflow. We will prescribe constant pressures $P(r, 0) = P_{in}$ and $P(r, L) = P_{out}$, where $P_{out}$ is taken to be zero. Now appears the question whether the transverse pressure profile in a fully developed turbulent flow is constant. In the following, we prove that it indeed is. We use the momentum conservation equations for turbulent flow (Equations 2.54 - 2.55). Written in expanded form (without tensor notation) in three dimensions, the $y$-momentum equation reads:
\[
\rho \frac{DV}{Dt} = -\frac{\partial P}{\partial y} + \frac{\partial}{\partial x} \left[ (\mu + \mu_T) \left( \frac{\partial V}{\partial x} + \frac{\partial U}{\partial y} \right) \right] + \frac{\partial}{\partial y} \left[ (\mu + \mu_T) \left( \frac{\partial V}{\partial y} + \frac{\partial V}{\partial y} \right) \right]
+ \frac{\partial}{\partial z} \left[ (\mu + \mu_T) \left( \frac{\partial V}{\partial z} + \frac{\partial W}{\partial y} \right) \right],
\]

(5.22)

where \( U \) and \( x \) are associated with the direction of the flow, while \( V, W, y, \) and \( z \) correspond to directions perpendicular to the flow. In a fully developed flow through a duct with constant shape cross-section the transverse velocity components vanish (that is \( V = W = 0 \)). Thus, we are left with the following equation:

\[
0 = -\frac{\partial P}{\partial y} + \frac{\partial}{\partial x} \left[ (\mu + \mu_T) \frac{\partial U}{\partial y} \right].
\]

(5.23)

The turbulent viscosity, \( \mu_T \), does not depend on \( x \) in a fully developed flow, therefore we can take it out of the \( x \) derivative. We also exchange the order of the \( x \) and \( y \) derivatives to obtain

\[
0 = -\frac{\partial P}{\partial y} + (\mu + \mu_T) \frac{\partial U}{\partial y} \frac{\partial U}{\partial x},
\]

(5.24)

where the streamwise velocity is not changing with \( x \), so the \( x \)-derivative of \( U \) will vanish \((\frac{\partial U}{\partial x} = 0)\), which implies that the pressure gradient (in the \( y \) direction) also vanishes:

\[
\frac{\partial P}{\partial y} = 0.
\]

(5.25)

In a similar way (by using the \( z \)-momentum equation) it can be proved that

\[
\frac{\partial P}{\partial z} = 0.
\]

(5.26)

Therefore

\[
\frac{\partial P}{\partial r} = 0,
\]

(5.27)

and

\[
P(r) = \text{constant},
\]

(5.28)

which entitles us to use a constant pressure profile as a boundary condition.

### 5.3.3 Meshing and Solution Procedure

COMSOL Multiphysics\textsuperscript{TM} is capable of discretising the computational domain automatically, without (or with little) user intervention. Both structured and unstructured grids are supported. For the generation of unstructured grids the Delaunay triangulation algorithm is used.

Although our computational domain is very regular, which encourages the use of structured meshes, it was decided to use unstructured grids for the discretisation step. As was already mentioned, the simulations described in this chapter serve the purpose of validating the chosen turbulence models, constants, boundary conditions and discretisation grids, so that they can be applied (with adjustments) to the more complicated case of fully
developed turbulent flow in periodically corrugated pipes. Unstructured grids will have to be used for corrugated pipes, hence they are also used here for non-corrugated pipes.

The meshes used in the simulation are shown in Figures 5.6 and 5.7. The solution procedure consists of three steps:

- Solve the model on the coarse mesh, shown in Figure 5.6.
- Refine the mesh to obtain the one shown in Figures 5.7.
- Solve the model on the refined mesh, using the coarse mesh solution as an initial guess.

This strategy proved to be faster than immediately solving the model on the refined mesh. After the solution was obtained it was additionally checked by refining the mesh once again and comparing the new solution to the previous one. The difference between them was less than 2%. A sample solution resulting from the simulations is shown in Figure 5.8.

5.4 Smooth Wall Validation

To validate the simulation of turbulent flow in pipes with smooth walls, the general simulation setup described in the previous section is used. At the boundary representing the
(smooth) wall of the pipe the standard BCs derived from the law of the wall are used (see Chapter 4). To compare the simulation results to the experimental data, a postprocessing procedure is performed at the end of each simulation. The following values are computed:

Average velocity

\[ V_{avg} = \frac{1}{\pi R^2} \int_0^R V \cdot 2\pi r dr, \]  

(5.29)

Reynolds number

\[ Re = \frac{2RV_{avg}}{\nu}, \]  

(5.30)

Friction factor

\[ f = \frac{2R\Delta P}{0.5\rho V_{avg}^2 L}, \]  

(5.31)

where \( \Delta P \) is the prescribed difference in pressure between in- and outflow, \( V_{avg} \) is the computed average velocity, and \( L \) is the length of the computational domain (in the axial direction).

It is worth mentioning here, that because of the way the law of the wall is used as a boundary condition in COMSOL Multiphysics\textsuperscript{TM}, the computational domain does not include the whole physical domain. Specifically, it does not include the thin layer near the wall where the velocity profile is given by the law of the wall (the shaded area in Figure 4.1). Therefore, in Equation 5.29 above, the radius \( R \) should be understood as \( R = R' + y_p \), where \( R' \) is the radius of the pipe in the simulations. In the computations the following values have been used: \( R = 0.2m \), \( L = 0.02m \).

![Friction factor computed with different turbulence models](image)

Figure 5.9: Computed and measured friction factors for smooth pipes \((e/D = 0)\).

Simulations were performed for a wide range of Reynolds numbers (from \( 10^4 \) up to \( 10^8 \)), using the two turbulence models \((k - \omega \) and \( k - e)\). For each model, two different boundary conditions at the pipe wall have been used. First, simulations were performed with the constant \( B = 5.5 \) in the law of the wall (see e.g. Equation 5.10), this being the value hard-coded in the boundary condition in COMSOL Multiphysics\textsuperscript{TM}. Then this
value was modified to 5.0 (as given in most of the literature on turbulent flows) by defining a custom law of the wall to be used as a boundary condition. Thus, we end up with four computed friction factors (for each Reynolds number), plus the friction factor measured experimentally (taken from the Moody diagram). These are shown in Figure 5.9. As can be seen from the plot, in the range of (relatively) low Reynolds numbers \((10^4 \text{ to } 10^5)\) the computed values are far from the measured friction factor. The \(k - \epsilon\) model seems to perform better in this range of \(Re\). For higher Reynolds numbers, the computed values follow closely the measured value; for \(Re > 5 \times 10^5\) the relative error is less than 0.04. Now the \(k - \epsilon\) and \(k - \omega\) models perform equally well and the choice of \(B\) in the boundary condition becomes important. As seen from Figure 5.10 (showing the plot from Figure 5.9 zoomed around \(Re = 10^6\)), for \(Re \leq 7 \times 10^5\) simulations with \(B = 5.0\) give closer agreement with the measured friction factor, while for \(Re > 7 \times 10^5\) the friction factor is better predicted by the simulations with \(B = 5.5\). This could be the reason why different authors give slightly different values for \(B\) in the law of the wall; its choice depends on the Reynolds number characteristic to the flow.

5.5 Rough Wall Modeling

The asperities on the wall of a bounded flow become important when their size is comparable to the thickness of the laminar sublayer. To model a turbulent wall-bounded flow in which the wall roughness effects are considered to be significant, they (roughness effects) can be included through the law of the wall modified for roughness. If \(e\) is the equivalent height of the asperities, the law of the wall is given by the relation (see e.g. [27]):

\[
\frac{U}{u_*} = \frac{1}{\kappa} \ln \left( \frac{y_p}{e} \right) + 8.5
\]  

(5.32)
5.5 Rough Wall Modeling

Figure 5.11 shows the friction factors computed using the law above. We observe that this law provides a good prediction of the friction factor only for Reynolds numbers higher than $10^6$ and is totally wrong for Reynolds numbers lower than $10^5$. To get a good approximation for the whole spectrum of Reynolds numbers, we need to combine the law for smooth walls with the law for rough walls.

Experiments in roughened pipes and channels indicate that the mean velocity distribution near rough walls, when plotted in the usual semi-logarithmic scale, has the same slope $(1/\kappa)$ but a different intercept (additive constant $B$ in the log-law). Thus, the law-of-the-wall for mean velocity modified for roughness has the form [12]

$$\frac{U}{u_*} = \frac{1}{\kappa} \ln\left(\frac{\rho u_* y p}{\mu}\right) + B^*, \quad (5.33)$$

where $u^* = C_{1/4} \kappa^{1/2}$ and $B^*$ is a roughness function that quantifies the shift of the intercept due to roughness effects.

For a sand-grain roughness and similar types of uniform roughness elements, however, $B^*$ has been found to be well-correlated with the nondimensional roughness height, $e^+ = \rho e u^*/\mu$, where $e$ is the physical roughness height. Analyses of experimental data show that the roughness function, $B^*$, is not a single function of $e^+$, but takes different forms depending on the $e^+$ value. It has been observed that there are three distinct regimes:

- Hydrodynamically smooth ($e^+ < e_1^+$)
- Transitional ($e_1^+ < e^+ < e_2^+$)
- Fully rough ($e^+ > e_2^+$)

where $e_1^+ \sim 3...5$ and $e_2^+ \sim 70...90.$
5.5 Rough Wall Modeling  

According to the data, roughness effects are negligible in the hydrodynamically smooth regime, but become increasingly important in the transitional regime, and take full effect in the fully rough regime.

In our case, the whole roughness regime is subdivided into the three regimes, and the formulas proposed by Ioselevich and Pilipenko in [12] are adopted to compute the roughness function, $B^*$, for each regime. Adopting the following expression for $B^*$:

$$B^* = B + \theta(8.5 - B - \frac{1}{\kappa} \ln e^+)$$  \hspace{1cm} (5.34)

the case of hydrodynamic smoothness corresponds to $\theta = 0$ ($e^+ < e_1^+$), while the case of full roughness corresponds to $\theta = 1$ ($e^+ > e_2^+$). The concrete form of the function $\theta = \theta(e^+)$ for $e_1^+ < e^+ < e_2^+$ is obtained in [12] from the analysis of experimental data. The following approximation is proposed for $\theta$:

$$\theta = \sin\left(\frac{\pi \ln(e^+/e_1^+)}{2 \ln(e_2^+/e_1^+)}\right).$$  \hspace{1cm} (5.35)

The values for $e_1^+$ and $e_2^+$ recommended by the authors are 2.25 and 90, respectively. These values were used in the present investigation. Thus we have:

- For the hydrodynamically smooth regime ($e^+ < 2.25$):
  $$\theta = 0.$$  \hspace{1cm} (5.36)

- For the transitional regime ($2.25 < e^+ < 90$):
  $$\theta = \sin\left[0.4258(\ln e^+ - 0.811)\right].$$  \hspace{1cm} (5.37)

- In the fully rough regime ($e^+ > 90$):
  $$\theta = 1.$$  \hspace{1cm} (5.38)

In COMSOL MultiphysicsTM, given the roughness parameters, the roughness function $B^*(e^+)$ is evaluated using 5.34 and the corresponding formula for $\theta$ (Equation 5.36, 5.37, or 5.38). The modified law of the wall in Equation 5.33 is then used to evaluate the velocity at the wall. This is implemented by defining the following global expressions:

```plaintext
eps = 0.01  \% parameter for heaviside step funct. flc2hs( , )
B = 5.5  \% the constant B in the law of the wall
dey = 0.002  \% absolute height of asperities
Kplus = dey*dwplus_chns/dw_chns  \% scaled roughness height
theta = flc2hs(Kplus-90,eps) + (flc2hs(Kplus-2.25,eps)-
    flc2hs(Kplus-90,eps))*sin(0.4258*(log(Kplus)-0.811))
Bstar = B+theta*(8.5-B-log(Kplus)/kappa_chns)
logfun = log(dwplus_chns)/kappa_chns+Bstar  \% modified log-law
```

Another approach for rough wall modeling, the viscosity adjustment method, is presented in Appendix A.
5.6 Rough Wall Validation

The model setup process repeats the one described in Section 5.3. The only difference is that the law of the wall is modified as described in the previous section. Equations 5.29-5.31 are used to compute the average velocity, the Reynolds number and the friction factor respectively.

Figure 5.12 shows the results of the simulations. They were obtained by varying two parameters:

- Pressure, \( P = \{0.1, 1, 10, 100, 1000, 10000\} \) Pa
- Relative roughness, \( e/D = \{0.0005, 0.001, 0.005, 0.01, 0.05\} \)

The pressure variation determines the variation of the Reynolds number, while the change of relative roughness generates a set of distinct curves on the diagram.

Figure 5.12: Computed (dotted lines) and measured (solid lines) friction factors for flow with rough walls using a combined law of the wall, Equation 5.33.

Figure 5.12 shows the results of the computations. Dotted lines indicate the computed friction factor, while solid lines correspond to the friction factor obtained from experiments (according to the Moody diagram). There is an excellent agreement of measured and computed values for \( Re > 10^6 \). In the transitional regime (\( 10^4 < Re < 10^5 \)), however, the friction factor seems to be underpredicted by our model. It can be noticed that our computations resemble the measurements of J. Nikuradse [17], shown in Figure 5.13. Nikuradse’s diagram is in essence identical to the Moody diagram, with a small difference in the transitional regime. This difference is caused by the fact that the transition from hydraulically smooth conditions at small Reynolds numbers to complete roughness at large Reynolds numbers occurs much more gradually in commercial rough pipes (used by Moody) than in artificially roughened pipes (used by Nikuradse).
5.6 Rough Wall Validation

Our computations resemble the data from Nikuradse, because the latter was used by Ioselevich and Pilipenko [12] to fit the combined law of the wall (Equations 5.33, 5.34, 5.36, 5.37, 5.38).

Figure 5.13: Friction factor measured by Nikuradse [17].
Chapter 6

Friction Factor Computations for Corrugated Pipes

We begin with a description of the characteristics of turbulent flow in corrugated pipes (e.g. new phenomena such as flow separation). Then a simulation of a fully developed turbulent flow in a corrugated pipe is set up. The computational domain, boundary conditions and grid generation are described. Finally, the model is used to estimate the influence of fabric roughness on the friction factor.

6.1 Characteristics of Flow in Corrugated Pipes

As it was explained in the Introduction (Chapter 1), flexible ducts have a specific structure, shown in Figure 6.1. The steel spiral, which gives strength and determines the shape of the pipe, causes a periodic corrugation inside the duct. The flow inside corrugated pipes has some important properties which are not characteristic to flows in conventional pipes. One of the expected effects that corrugation might have on the flow is that the transition from laminar to turbulent flow will occur at lower Reynolds numbers (so, at $Re < 2000$). Nishimura and colleagues [18], Russ and Beer [22]-[23], and Yang and colleagues [28] investigated the transitional flow characteristics in corrugated ducts. Indeed, they reported the laminar-turbulent transition to occur at very low Reynolds numbers compared with conventional ducts.

![Flexible duct](image)

Figure 6.1: Flexible duct.

Another important characteristic of flow in corrugated pipes is the possible presence of adverse pressure gradients and, as a result, the appearance of flow separations. The
pressure gradient (be it favourable or adverse) is one of the factors that influences a flow immensely. The shear stress caused by viscosity and turbulence has a retarding effect upon the flow. This effect can however be overcome if there is a negative pressure gradient applied to the flow. A negative pressure gradient is termed a favourable pressure gradient. Such a gradient drives the flow. A positive pressure gradient has the opposite effect and is termed the adverse pressure gradient. Fluid might find it difficult to overcome an adverse pressure gradient. Sometimes, it is said that the fluid has to climb the pressure hill.

One of the severe effects of an adverse pressure gradient is its tendency to separate the flow. Let us consider the flow past a curved surface as shown in Figure 6.2. The geometry of the surface is such that we have a favourable gradient in pressure to start with and up to a point $P$. The negative pressure gradient will counteract the retarding effect of the shear stress (which is due to viscosity) in the boundary layer. For the geometry considered we have an adverse pressure gradient downstream of $P$. After having passed the point $P$, the flow begins to be retarded by the adverse pressure gradient. This effect is felt more strongly in the regions close to the wall where the momentum is lower than in the regions near the free stream. As indicated in the figure, the velocity near the wall reduces and the boundary layer thickens. A continuous retardation of flow brings the velocity (as well as its gradient and the wall shear stress) near the point $S$ on the wall to zero. From this point onwards the flow reverses and a region of recirculating flow develops. The shear stress becomes negative. We see that the flow no longer follows the contour of the body. We say that the flow has separated. The point $S$ where the shear stress is zero is called the point of separation.

Depending on the flow conditions and surface geometry the recirculating flow terminates and the flow may become reattached to the body. A separation bubble is formed. There are a variety of factors that could influence this reattachment. The pressure gradient may be now favourable due to body geometry and other factors.

![Figure 6.2: Separation of flow over a curved surface.](image)

### 6.2 Simulation Setup

The objective in this section is to set up a model in COMSOL Multiphysics™ which will allow us to perform a set of simulations and compute the friction factor depending on parameters such as roughness of the hose wall. Before we perform the simulations, a proper turbulence model has to be chosen ($k - \epsilon$, $k - \omega$, or both). As we have seen in the previous section, because of the curved surfaces present in the geometry, we expect
to encounter separated flow. Prediction of separated flows is an "Achilles heel" for many turbulence models, therefore simulation of turbulent flow over a backward facing step (a standard test problem for separated flow) is performed using \( k-\epsilon \) and \( k-\omega \) turbulence models. Results, presented in Appendix C, show that although both models underpredict the reattachment length, the values given by the \( k-\epsilon \) model are closer to experimental data. Therefore, we choose the \( k-\epsilon \) model for our computations. After we have chosen the model, we also need to determine the computational domain, the boundary conditions, generate a mesh, and solve the model. All these steps are described in the following subsections.

### 6.2.1 Computational Domain Geometry

In general, fully developed turbulent flow in a corrugated pipe is of three-dimensional nature. For our particular structure (Figure 6.1), it can be simplified to two-dimensional because of axial symmetry. In other words, it only depends on the distance from the pipe’s centerline, \( r \), and the position along the pipe, \( x \).

![Figure 6.3: Computational domain.](image)

Figure 6.3 shows the computational domain for the simulated pipe. It includes one period of the periodically corrugated pipe. At the left and at the right it is bounded by the wall and by the symmetry axis, respectively, while the inflow/outflow boundaries are the ones located at the top and bottom of the domain in Figure 6.3. Since we are investigating a fully developed flow, we are not interested in any temporal behaviour, so
the steady Navier-Stokes equations are used. To include the effects of turbulence, two additional equations (Equations 2.56 and 2.57 for the $k - \epsilon$ model, and Equations 2.61 and 2.62 for the $k - \omega$ model) are added to the model, as discussed in Chapter 2.

6.2.2 Boundary Conditions

As can be seen from Figure 6.3, our computational domain has five distinct boundaries at which boundary conditions have to be prescribed. As explained above, we use a 2D axisymmetric model. Thus, at one of the boundaries parallel to the flow the Axial Symmetry boundary condition is prescribed. This boundary coincides with the centerline of the pipe. The other boundary parallel to the flow coincides with the wall of the pipe, which consists of two different materials: a flexible hose made of fabric (numbers 4 and 5 on the figure) and a steel spiral (numbers 6 and 7). The spiral, made of steel, is considered a smooth surface. Therefore, the standard law of the wall is used there as a boundary condition. The part made of fabric, on the other hand, is considered to be rough, so the law of the wall modified for roughness (as described in Section 5.5) is used.

The boundaries normal (perpendicular) to the flow represent inflow and outflow boundaries. The duct is considered to be infinitely long in the main flow direction. Far from the duct entrance, the flow will be periodically fully developed [19] because of the periodic positioning of the corrugations. In a periodic fully developed flow, the velocity repeats itself at corresponding axial locations in successive cycles. Now it becomes clear why it is enough to confine the computational domain to a single period (cycle) of the corrugation.

From the above statements we have the following periodic boundary conditions for velocity:

\begin{align}
U(r, 0) &= U(r, L), \\
V(r, 0) &= V(r, L).
\end{align}

(6.1)

(6.2)

where $L$ is the length of the computational in the axial direction (set as $L = 0.12$ in simulations)

Also, the turbulence-related quantities ($k$, and $\epsilon$ or $\omega$) are assumed to have the same values at inflow and outflow:

\begin{align}
k(r, 0) &= k(r, L), \\
\epsilon(r, 0) &= \epsilon(r, L), \\
\omega(r, 0) &= \omega(r, L).
\end{align}

(6.3)

(6.4)

(6.5)

Now remains the question of boundary conditions for the pressure at the inflow/outflow boundaries. In the periodic fully developed regime, the pressures of cyclically corresponding locations decrease linearly in the downstream direction [8]. Thus, the pressure $p$ can be expressed as

\begin{equation}
p(x, r) = -\beta x + p'(x, r),
\end{equation}

(6.6)

where $\beta$ is the mean pressure gradient and $p'(x, r)$ is the periodic component of the pressure (behaves in a periodic manner from cycle to cycle). The term $-\beta x$ represents the nonperiodic pressure drop that takes place in the flow direction. Then, the pressures at the inflow and outflow can be written
6.2 Simulation Setup

\[ p(x_{in}, r) = -\beta x_{in} + p'(x, r), \]  
\[ p(x_{out}, r) = -\beta x_{out} + p'(x, r). \]  

(6.7)  
(6.8)

We subtract the second equation from the first equation and denote by \( L \) the period of the corrugation \( (L = x_{out} - x_{in}) \)

\[ p(x_{in}, r) - p(x_{out}, r) = \beta (x_{out} - x_{in}) = \beta L. \]  

(6.9)

Keeping in mind that \( \beta \) is the mean pressure gradient, we have that \( \beta L = \Delta P \), with \( \Delta P \) being the pressure difference between the inflow and outflow boundaries. Thus, we obtain the desired periodic boundary condition for \( p \):

\[ p(x_{in}, r) = p(x_{out}, r) + \Delta P. \]  

(6.10)

The use of periodic boundary conditions represents a very effective method for simulating periodically fully developed flow and brings an important decrease of computational costs, which in turn facilitates the use of finer meshes.

6.2.3 Meshing, Solution Procedure, Discussion of a Typical Solution

As already mentioned, COMSOL Multiphysics\textsuperscript{TM} provides tools for automatic mesh generation. Two strategies may be distinguished for mesh generation:

- a priori mesh generation
- adaptive mesh generation
Methods in the first of these categories do not use any information of the solution to be calculated. In other words: the mesh is constructed before the solution is calculated. Nevertheless, information about previously calculated solutions and characteristics of the problem may be used to construct the mesh. Methods in the second category are characterized by the fact that the construction of the mesh and the calculation of the corresponding solution are performed simultaneously. Adaptive mesh generation identifies the high-activity regions that require a high resolution (by estimating the errors) and produces an appropriate mesh.

To perform the simulations the adaptive solver included in COMSOL Multiphysics™ was used. A coarse mesh, as shown in Figure 6.4, was used as a starting point. It was adaptively refined by the solver to finally arrive at the mesh shown in Figure 6.5. We notice that the regions around the corrugation and near the wall require much higher resolution than the area near the center of the pipe.

Figure 6.6 shows the convergence of the solution when using the adaptive solver. First, the model is solved on the coarse mesh. Then the errors are estimated and the mesh is refined at the locations with larger errors. The model is solved again on the new mesh. The process continues until the errors are smaller than a certain threshold, or the maximum number of mesh refinements is reached.

A typical solution for the case of periodically fully developed turbulent flow in a corrugated pipe (at $Re \approx 10^6$) is displayed in Figure 6.7. The coloured surface corresponds to the value of pressure, arrows indicate the direction and the magnitude of the velocity field, while the green lines near the wall are the streamlines of the flow. As expected, there is a region of higher pressure before the bump (corrugation), followed by a region of lower pressure. By doing some computations for different Reynolds numbers it became clear that the low-pressure region is located after the bump in flows at low Reynolds number, and it moves towards the top of the bump as the Reynolds number gets higher. There is an adverse pressure gradient on the top of the bump. However, due to the large
6.3 Influence of Fabric Roughness on the Friction Factor

Figure 6.6: Convergence vs. iteration number.

enough velocity, the flow has enough momentum to overcome it and keep flowing in the mainstream direction. Not the same is the situation for the flow immediately behind the bump; the velocity is lower and it keeps decreasing until the fluid begins to flow in the opposite direction. Thus, a separation of the flow occurs and a recirculation zone (a bubble) is formed.

The transverse pressure profile is shown in Figure 6.8. It is no longer constant as it was the case for a non-corrugated pipe. The pressure is almost constant near the centerline and increases towards the corrugated wall.

6.3 Influence of Fabric Roughness on the Friction Factor

One of the goals of this investigation was to answer the question whether the type of the fabric (which is wrapped around the steel spiral) has a strong influence on the friction factor for the resulting pipe. In order to do this, a new type of boundary condition based on a "combined" law of the wall was implemented in COMSOL Multiphysics™. The "combined" law of the wall represents an approach for roughness modeling and is explained in Section 5.5. So that we can believe the results of simulations with models using the new boundary condition, the flow in a non-corrugated (conventional) pipe, for which experimental data exists, was used as a validation test. The validation procedure and its results are described in Section 5.6.

Now, that we know we can trust our boundary conditions, we can investigate the influence of fabric roughness on the friction factor. Simulations were performed for a set of values for relative roughness of the fabric, $\epsilon/D = \{0, 0.0125, 0.025, 0.05\}$. The curve of computed friction factors is shown Figure 6.9. Two observations can be made here:

- The friction factor increases as the relative roughness increases (this is what we
6.3 Influence of Fabric Roughness on the Friction Factor

Figure 6.7: Typical solution. Coloured surface - pressure, arrows - velocity fields, green lines - streamlines.

• The slope of the function is extremely small

The difference between the two limiting cases $e/D = 0$ and $e/D = 0.05$ is less than 10%. This entitles us to state that the roughness of the fabric has a negligible influence on the friction factor. Which in turn means that there is no point in looking for new materials which would keep the properties of the older ones (such as non-inflammable, resistive to high pressures, resistive to extreme temperatures, etc.) but would have a lower roughness, because the decrease in the friction factor is expected to be very small.
6.3 Influence of Fabric Roughness on the Friction Factor

Figure 6.8: Pressure profile along the radial direction at $x = 0$.

Figure 6.9: Friction factor vs. fabric roughness.
Chapter 7

Conclusions

The performance of two-equation turbulence models was assessed. Two most popular representatives of this class of turbulence models were chosen to be used: the $k-\epsilon$ and the $k-\omega$ models. They have been tested for simple pipe flow and for a more complicated case of flow over a backward-facing step. The latter test was needed in order to assess the ability of the models to predict flow separation, both qualitatively and quantitatively. These simulations have shown that both models perform reasonably well in the case of pipe flow, with a small advantage of the $k-\epsilon$ model over the $k-\omega$ model at low Reynolds numbers ($10^4 < Re < 10^5$). Also, it has been shown that the experimental results are closer reproduced if the constant $B$ in the law of the wall has a value of 5.0 for $Re \leq 7 \times 10^5$, and 5.5 for $Re > 7 \times 10^5$. The tests concerning flow over a backward-facing step have confirmed the tendency of two-equation models to underpredict the reattachment length. However, the results obtained from the $k-\epsilon$ model are closer to the measurements. For this reason, the $k-\epsilon$ model was chosen for the simulation of (fully developed turbulent) flow in corrugated pipes, as we expect to encounter flow separation behind the corrugations.

The effects of wall roughness have been modeled by implementing a new boundary condition based on a "combined" law of the wall. The flow in a non-corrugated (conventional) pipe with rough walls, for which experimental data exists, was used as a test problem. The results of the test reproduce approximately the Moody diagram (except for a small range of $Re$ around $10^5$) and are in close agreement with Nikuradse’s experimental data.

Being able to trust our models and boundary conditions, the question whether the type of the fabric used has a strong influence on the friction factor was investigated. The performed computations have shown that the roughness of the fabric has a negligible influence on the friction factor. This in turn means that no considerable advantage is gained by replacing a rough fabric by a smooth one. The bumps created by the steel spiral cause most resistance and considerable drag reduction can only be obtained by optimising their shape.
Appendix A

Viscosity Adjustment Approach for Roughness Modeling

A.1 Approach for Roughness Modeling

When a turbulence transport model, such as $k - \epsilon$, is used in numerical simulations there must be suitable boundary conditions for the turbulence quantities (i.e., turbulent kinetic energy $k$ and turbulent dissipation $\epsilon$) that reflect the roughness of the boundary. Typically, these boundary conditions are set using so called ‘wall functions’. That is, it is assumed that a logarithmic velocity profile exists near a wall, which can be used to compute an effective shear stress at the wall. One does this by fitting the log profile to the computed tangent velocity near the wall, $U$, at the distance of that velocity component from the wall, $y_p$. For a smooth wall the equation is,

$$U = u_\ast \left[ \frac{1}{\kappa} \ln \left( \frac{\rho u_\ast y_p}{\eta} \right) + 5.0 \right].$$

(A.1)

In this equation $\kappa$ is the von Karman constant and the equation is solved for the wall shear stress velocity, $u_\ast$. The $u_\ast$ value is then used to define wall boundary conditions for the turbulent variables $k$ and $\epsilon$. For the $k - \epsilon$ model the boundary conditions are:

$$k = \frac{u_\ast^2}{\sqrt{C_\mu}}, \quad \epsilon = \frac{u_\ast^3}{\kappa y_p},$$

(A.2)

where $C_\mu = 0.09$ is one of the five parameters in the $k - \epsilon$ model.

For rough walls the same procedure is used, but the logarithmic function has a slightly different form,

$$U = u_\ast \left[ \frac{1}{\kappa} \ln \left( \frac{y_p}{e} \right) + 8.5 \right].$$

(A.3)

The incorporation of rough-wall boundary conditions into COMSOL MultiphysicsT$M$ is accomplished by assuming that roughness makes an additional contribution to the molecular viscosity $\eta$ [25], forming an effective viscosity defined as

$$\mu_{eff} = \mu + \rho \cdot a \cdot u_\ast \cdot e.$$  

(A.4)

Here $a$ is a constant and $e$ is the roughness value input for the solid surface (in meters). The idea behind this proposition is that roughness elements introduce perturbations in
the flow characterized by \( u_s \) and \( e \) that increases the transfer of momentum between the fluid and the boundary. To see the convenience of the \( \mu_{\text{eff}} \) expression, we substitute it in place of \( \mu \) into the smooth-wall logarithmic profile (Equation A.1),

\[
U = u_s \left[ \frac{1}{\kappa} \ln \left( \frac{\rho \cdot u_s \cdot y_p}{\mu + \rho \cdot a \cdot u_s \cdot e} \right) + 5.0 \right]. \tag{A.5}
\]

When roughness is zero and \( \mu_{\text{eff}} \) is simply equal to the molecular viscosity the logarithmic velocity profile is the correct expression for a smooth wall. However, when the roughness contribution to \( \mu_{\text{eff}} \) is much larger than the molecular value the logarithmic profile reduces to something that looks like the rough wall expression. If we define \( a = 0.246 \), then this becomes the accepted expression for rough walls, with the constant 5.0 replaced by 8.5. Thus, the use of an effective viscosity that is the sum of the molecular viscosity and a roughness contribution at a rough-wall boundary offers a convenient way to make a continuous transition between smooth and rough wall boundary conditions.

### A.2 Model Setup in COMSOL Multiphysics™

As it was explained in the previous section, in order to model wall roughness in COMSOL Multiphysics™ we have to change dynamically the effective value of fluid viscosity at the wall. First, a step function is defined which will indicate whether the original viscosity or a modified viscosity must be used:

\[
\text{step}(r) = \text{heaviside}(r - (R - \text{delta})). \tag{A.6}
\]

However, when we provide a function, coefficient or material property as a step function to COMSOL Multiphysics™, we may experience convergence problems. This is because COMSOL Multiphysics™ expects the solution to be continuous. For time-dependent problems, the time-stepping algorithm can run into problems. For stationary problems (as in our case), there can be mesh resolution issues like over- and undershooting of the solution due to infinite flux problems [5].

In order to overcome the above difficulties, we can use smoothed switch functions that emulate steps. This serves the purpose of enhancing numerical reliability and convergence.

Therefore, the function \texttt{flc1hs}, available in COMSOL Multiphysics™, is used. It represents a smoothed Heaviside function with a continuous first derivative. \( y = \text{flc1hs}(x, \text{scale}) \) approximates the logical expression \( y = (x > 0) \) by smoothing the transition within the interval \(-\text{scale} < x < \text{scale}\).

Using the ‘smoothed’ step function, the effective viscosity is computed as:

\[
\mu_{\text{eff}} = \mu + \rho \cdot a \cdot u_s \cdot \text{rough} \cdot \text{step}(r) \tag{A.7}
\]

Below the definitions of all global expressions are listed:
A.3 Results

Values used in the computations:
Relative roughness, \( \frac{\epsilon}{D} = \{0, 0.01, 0.03, 0.05\} \)
Pressure, \( P = \{0.1, 1, 10, 100, 1000, 10000\} \) Pa.

A.4 Effects of Mesh Refinement

By computing the friction factors using different meshes, we get the following results:

<table>
<thead>
<tr>
<th>( \frac{\epsilon}{D} )</th>
<th>0</th>
<th>0.01</th>
<th>0.03</th>
<th>0.05</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse mesh</td>
<td>0.010</td>
<td>0.029</td>
<td>0.039</td>
<td>0.049</td>
</tr>
<tr>
<td>Fine mesh</td>
<td>0.010</td>
<td>0.029</td>
<td>0.047</td>
<td>0.071</td>
</tr>
</tbody>
</table>

Thus, it seems that the size of the mesh has a greater impact for higher relative roughness. However by further refining the mesh, eventually a negative(!) velocity is obtained in the case of non-smooth walls.
Figure A.1: Friction factor computed from simulations.
Appendix B

Influence of Wall Region Thickness on the Solution

Note: Simulations performed here are using the same mesh for all the computations. Because of this, the results turned out to be far from the truth. Later, using an adaptive solver (with adaptive mesh refinement), much better results have been obtained (see Section 4.3 and Figure 4.2). For the sake of completeness of the report, the old results are still included in the report.

B.1 Model Setup

Summary of the model:

<table>
<thead>
<tr>
<th>Model</th>
<th>Averaged Navier-Stokes + 2 eqns for $k$ and $\epsilon$</th>
</tr>
</thead>
<tbody>
<tr>
<td>BC at centerline</td>
<td>Symmetry</td>
</tr>
<tr>
<td>BC at in- and outflow</td>
<td>Periodicity for $u$, $v$, $k$ and $\epsilon$; Prescribed pressure $P$</td>
</tr>
<tr>
<td>BC at the wall</td>
<td>(Modified) log-function for smooth wall</td>
</tr>
<tr>
<td>Solver type</td>
<td>Parametric</td>
</tr>
<tr>
<td>Linear system solver</td>
<td>PARDISO and UMFPACK (for rough walls)</td>
</tr>
</tbody>
</table>

Mesh used:

B.2 Results

Values used in the computations:
Relative roughness, $\frac{\varepsilon}{D} = \{0, 0.01\}$.
Wall region thickness, $y_p^+ = \{2.5, 5, 10, 20, 40, 80, 160, 320, 640, 1280\}$.
Pressure, $P = \{0.1, 10, 1000\}$ Pa.
Composed friction factor, \( P = 0.1 \text{Pa}, \, Re \approx 1.2 \times 10^4, \, e/D = 0.01. \)
Computed friction factor, $P = 10$ Pa, $Re \approx 1.4e5$, $e/D = 0$.

Computed friction factor, $P = 10$ Pa, $Re \approx 1.4e5$, $e/D = 0.01$. 
B.3 Conclusions

Analyzing the results, two cases can be distinguished. For rough walls, the friction factor is always maximal for small values of \( y^+ \) (around 10). However, in the case of perfectly smooth walls, the friction factor depends strongly on the Reynolds number. Thus, the higher the Reynolds number, the higher the values of \( y^+ \) for which the friction factor is maximal.

Computed friction factor, \( P = 1000 \text{ Pa}, \ Re \approx 1.8\times10^6, e/D = 0 \).

Computed friction factor, \( P = 1000 \text{ Pa}, \ Re \approx 1.8\times10^6, e/D = 0.01 \).
Also, it must be mentioned that the near-wall region was not included in the computation of average velocity (and, as a result, the computation of friction factor). Its inclusion might insignificantly change the results.
Appendix C

Performance of Two-Equation Models for Separated Flow Prediction

The backward facing step (Figure C.1) is a standard case for studying the performance and solution strategy of a turbulence model, when separated flow is expected to be present in the solution. In this case, flow is subjected to a sudden increase of the cross-section, resulting in a separation of flow starting at the point of expansion. Spatial variations in the velocity field cause a production of turbulence outside the wall region and its interaction with the mean flow influences the size of the separation zone. The size of the zone, or the re-attachment length, is one of the quantities that must be predicted accurately by a turbulence model. The Reynolds number and the expansion ratio are two numbers that can be used to characterize the flow. Existing experimental data indicates that the length of the re-attachment zone increases with the expansion ratio. The Reynolds number is calculated based on the step height and the inlet free stream velocity. The re-attachment length increases with Reynolds number until 1200, where it then decreases between $1200 < Re < 6600$, to then be relatively constant for higher Reynolds numbers, when the flow is in state of fully developed turbulence.

This example calculation treats the backward facing step for a flow configuration with an area expansion ratio of $3/2$. The step height, $h_{step}$, is chosen as the length scale, the inlet is set to $2h_{step}$, the initial channel length to $8h_{step}$, and the length of the channel after the expansion is set to $35h_{step}$ (see Figure C.1).

C.1 Performance of the $k − \omega$ Model

The $k − \omega$ model is used to compute the turbulent flow over a backstep geometry at $Re = 44000$. The following two tables give a summary of the computational model and the constants used in the example.

<table>
<thead>
<tr>
<th>Model Elements</th>
<th>Averaged Navier-Stokes + 2 eqns for $k$ and $\omega$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solver type</td>
<td>Quadratic (Lagrange element of order 2)</td>
</tr>
<tr>
<td>Linear system solver</td>
<td>Adaptive</td>
</tr>
<tr>
<td></td>
<td>UMFPACK</td>
</tr>
</tbody>
</table>
C.1 Performance of the $k-\omega$ Model

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density, $\rho$</td>
<td>1.23 kg/m$^3$</td>
</tr>
<tr>
<td>Dynamic viscosity, $\mu$</td>
<td>1.79e-5 Pa·s</td>
</tr>
<tr>
<td>Height of the step, $h_{step}$</td>
<td>0.0381 m</td>
</tr>
</tbody>
</table>

Figure C.1: Computational domain.

The geometry of the test problem is shown in Figure C.1. The boundaries are numbered and the condition at each boundary is given below:

**Boundary 1: Inflow**

\[
\begin{align*}
U &= U_0, \\
k &= 1.5I_T^2(U_0 \cdot U_0), \\
\omega &= C_{mu}^{-0.25} \left[1.5I_T^2(U_0 \cdot U_0)\right]^{0.5} / L_T
\end{align*}
\]

where $U_0 = (18.2, 0)^T$ m/s is the inflow velocity vector, $I_T = 0.054$ is the turbulent intensity scale, and $L_T = 0.07$ m is the turbulent length scale.

**Boundary 2-6: Logarithmic wall function**

\[
\begin{align*}
n \cdot U_p &= 0, \\
(\mu + \mu_T) \left[\nabla U_p + (\nabla U_p)^T\right] \cdot n &= \frac{\rho C_{mu}^{1/4} k_p^{1/2}}{\kappa \ln y_p^+ + B} U_p, \\
n \cdot \nabla k_p &= 0, \\
\omega_p &= \frac{\rho k_p}{\kappa y_p^+ \mu}
\end{align*}
\]

where $y_p^+ = 100$ is the thickness of the log layer.

**Boundary 7: Outflow**

\[
\begin{align*}
n \cdot U &= 0, \\
P &= 0, \\
n \cdot \nabla k_p &= 0, \\
n \cdot \nabla \omega_p &= 0.
\end{align*}
\]
C.2 Performance of the $k - \epsilon$ Model

The $k - \epsilon$ model is used to describe the turbulent flow over a backstep geometry at $Re = 44000$. The following two tables give a summary of the model and the constants used in the example.

<table>
<thead>
<tr>
<th>Model</th>
<th>Averaged Navier-Stokes + 2 eqns for $k$ and $\epsilon$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elements</td>
<td>Quadratic (Lagrange element of order 2)</td>
</tr>
<tr>
<td>Solver type</td>
<td>Adaptive</td>
</tr>
<tr>
<td>Linear system solver</td>
<td>UMFPACK</td>
</tr>
</tbody>
</table>

| Density, $\rho$    | 1.23 $kg/m^3$ |
| Dynamic viscosity, $\mu$ | 1.79e-5 $Pa \cdot s$ |
| Height of the step, $h_{\text{step}}$ | 0.0381 $m$ |

Figure C.2: The mesh generated by the adaptive solver around the expansion region.

The final mesh generated by the adaptive solver is shown in Figure C.2. We notice that the mesh is much finer at the point of expansion.

Figure C.3 shows the computed solution. The colored surface corresponds to $U$ (the $x$-component of the velocity), the black lines are the streamlines, and the white line is the contourline for $U = 0$. The location where the white line meets the bottom of the channel is the point of reattachment, $x_{ra}$. The dimensionless re-attachment length is computed as:

$$L_{ra} = \frac{x_{ra}}{h_{\text{step}}} = \frac{0.159}{0.0381} = 4.17$$  \hspace{1cm} (C.12)

C.2 Performance of the $k - \epsilon$ Model

The $k - \epsilon$ model is used to describe the turbulent flow over a backstep geometry at $Re = 44000$. The following two tables give a summary of the model and the constants used in the example.
Figure C.3: Plot of the solution around the expansion region. Colored surface - $U$ (x-velocity), black lines - flow streamlines, white line - contourline for $U = 0$.

The geometry of the model is the same as for the $k - \omega$ model (see Figure C.1). The boundary conditions for each boundary is given below:

**Boundary 1: Inflow**

\[
U = U_0 \\
k = 1.5I_T^2(U_0 \cdot U_0) \\
\epsilon = C_{mu}^{0.75} \left[1.5I_T^2(U_0 \cdot U_0)\right]^{1.5}/L_T
\]

where $U_0 = (18.2, 0)m/s$ is the inflow velocity vector, $I_T = 0.054$ is the turbulent intensity scale, and $L_T = 0.07m$ is the turbulent length scale.

**Boundary 2-6: Logarithmic wall function**

\[
n \cdot U_p = 0, \quad \mu + \mu_T \left[\nabla U_p + (\nabla U_p)^T\right] \cdot n = \frac{\rho C_{\mu}^{1/4} k_p^{1/2}}{\frac{1}{k} \ln y_p^+ + B} U_p, \\
n \cdot \nabla k_p = 0, \\
\epsilon_p = \frac{\rho C_{\mu} k_p^2}{\kappa y_p^+ \mu}
\]

where $y_p^+ = 100$ is the thickness of the log layer.
C.3 Discussion of Results

Boundary 7: Normal flow, pressure

\[
\begin{align*}
\mathbf{n} \cdot \mathbf{U} &= 0, \quad (C.20) \\
P &= 0, \quad (C.21) \\
\mathbf{n} \cdot \nabla k_p &= 0, \quad (C.22) \\
\mathbf{n} \cdot \nabla \epsilon_p &= 0. \quad (C.23)
\end{align*}
\]

The final mesh generated by the adapted solver is shown in Figure C.4. Again, the mesh is much finer at the point of expansion.

Figure C.4: The mesh generated by the adaptive solver around the expansion region.

Figure C.5 shows the solution. This time, the dimensionless re-attachment length is larger:

\[
L_{ra} = \frac{x_{ra}}{h_{step}} = \frac{0.216}{0.0381} = 5.67 \quad (C.24)
\]

C.3 Discussion of Results

We have obtained two solutions for the same flow by using two different turbulence models. Figure C.3 gives the solution of the \( k - \omega \) model and Figure C.5 - the solution of the \( k - \epsilon \) model. Both figures show that there is a recirculation zone after the backstep. However, the size of the recirculation zone is different for the two models. The dimensionless re-attachment length is 4.17 for the \( k - \omega \) model and 5.67 for the \( k - \epsilon \). This can be compared to results presented in [2]. There, the reattachment length is experimentally determined to be 7.1, while numerical results using the \( k - \epsilon \) model yielded 6.1. Thus, we see that
two-equation models do not perform very well when dealing with separated flow. When comparing the two models, it is obvious that the $k - \epsilon$ model predictions are closer to the experiments.


