Numerical simulation of the flow in a scour hole due to a translating jet.

F.J.C. Visscher

DELFt UNIVERSITY OF TECHNOLOGY
SECTION OF DREDGING ENGINEERING

version TU(final) Monday 7th March, 2016
Numerical simulation of the flow in a scour hole due to a translating jet.

F.J.C. Visscher
A thesis presented for the degree of
MSc. Offshore and Dredging Engineering

Monday 7th March, 2016
Version: TU(final)

Coordinators:

Dr.ir. G.H. Keetels
TU-Delft
Department Maritime & Transport Technology
Section Offshore & Dredging Engineering

ir. R.A. van der Hout
Van Oord Dredging & Marine Contractors bv
Department Engineering & Estimating

TU Delft
University of Technology
Faculty of Mechanical, Maritime & Materials Engineering

Van Oord
Marine ingenuity
Abstract

Dredging is a very important activity for mankind, for example deepening waterways or creating artificial islands. The Trailing Suction Hopper Dredger (TSHD) is an important dredging vessel, this vessel uses the so called drag head. It is common practice to use water jets in drag heads to create a more efficient dredging process. Some theoretical and empirical models exist which determine for example the van Rhee jet model which predicts the depth and width of the scour hole created by a impinging jet. The performance of these models is limited such that the production is not estimated correctly. Therefore a lot of effort is done to understand the underlying processes. This thesis project is therefore aiming to provide insight in these quantities using numerical simulations of the flow.

In the year 2008 Stichting Speurwerk Baggertechniek (SSB) performed experiments in the Deltares dredging laboratory in Delft (the Netherlands). These experiments consisted of applying a translating jet positioned horizontal above a sand bed, operating at high jet powers. These experiments were well documented and resulted in measuring data which were post processed by the SSB partners individually. However both the velocity distributions and the sand concentrations in the body where not measured. So a possibility to correlate the erosion velocities does not exist.

The results from the experiments are used in this thesis project to create the numerical simulations of the flow so the velocity distributions can be predicted. Further the scour hole formation can be checked with existing erosion theory. Well-documented and similar (to the flow in the scour hole) cases are sought and discussed which enable us to validate the software package used and validate its applicability on the flows studied in this project.

The plane wall jet is then simulated using 2 different turbulence models. Also several available wall functions, which are parameterization of the near wall flow in order to reduce calculation costs, are used. It is found that the two turbulence models give significantly different results. While trying to set up a simulation of the 3D jet in the scour hole it becomes clear that concessions have to be made due the complexity of erosion processes. Not only this forces to make compromises, but also the incompatibility of the desired turbulence model with the mixture flow solver in OpenFOAM forces us to run single phase simulations.

However after evaluation of the numerical simulations for the 3D jet in the scour hole, a similarity behavior of the velocities observed. This provides building stones for
creating a simple model to determine the velocities inside the scour hole. When this model is created it can be used to improve existing jet models to predict the scour hole formation and production process of the drag head. Finally this would help the Dutch dredging industry to maintain their leading position on the global market.
Preface

This thesis is the graduation project for the degree Master of Science for the study Offshore and Dredging Engineering at the Delft University of Technology. In this thesis it is investigated whether it is possible to predict velocities in a translating jet scour hole using computational fluid dynamics.

I would like to use this opportunity to express my gratitude towards all the persons that provided support during this project, especially the committee members from the university and Van Oord. For giving the good suggestions during the research traject I want to thank my supervisors Ronald van der Hout and Geert Keetels. On top of that the useful conservations with Michel de Nijs are also extraordinary appreciated because of preliminary path became clear in an early stage. Also I would like to thank Ronald van der Hout and Marcel van de Heuvel for their comments and suggestions how to create a complete and understandable report.

Finally, I would like to thank my family, friends, girlfriend and colleagues for their interest.

Confidentiality

The experiments, and their results, done by the SSB are confidential. This means that this thesis project is also strictly confidential.

Digitised Figures

In the report some data from other publications are converted from the raster images to vector images, most of this work is copyright protected material and therefore care is given to make the proper references to both author and publisher. If for some reason this is not done properly please contact me.
## Contents

Abstract ................................................................. i

Preface ................................................................. iii

1 Introduction ............................................................ 1
   1.1 General problem ................................................ 1
   1.2 Goal ............................................................. 4
   1.3 Approach ........................................................ 4

2 Traversing impinging jet on sand .................................. 5
   2.1 Introduction ..................................................... 5
      2.1.1 Van Rhee Model ......................................... 7
      2.1.2 BAGT-540 ................................................. 7
      2.1.3 SSB-P19 .................................................. 8
      2.1.4 Creating scour hole surfaces .......................... 8
      2.1.5 ‘Bottom’ and ‘Side’ ..................................... 8
      2.1.6 Erosion velocities (kinematic) ....................... 9
   2.2 Wall Jet .......................................................... 11
      2.2.1 Plane Wall Jet Similarity ................................ 12
      2.2.2 Equations ................................................ 13
      2.2.3 Experimental results ..................................... 13
      2.2.4 Found relations .......................................... 14
      2.2.5 Experimental work ....................................... 17
      2.2.6 Turbulent three-dimensional wall jet ............... 17
   2.3 Erosion and Settling ........................................... 18
      2.3.1 Bed growth .............................................. 20
      2.3.2 Settling and hindered settling ....................... 20
      2.3.3 Pick-up ................................................ 21

3 Computational Fluid Dynamics ....................................... 23
   3.1 Introduction ................................................... 23
   3.2 Basic Equations ................................................ 23
      3.2.1 Mass Conservation ...................................... 23

v
## CONTENTS

3.2.2 Linear Momentum .............................................. 24
3.2.3 Navier-Stokes Equations ................................. 25

3.3 Turbulence .......................................................... 25
3.3.1 Turbulent energy .............................................. 26
3.3.2 Turbulence intensity ......................................... 26
3.3.3 (Turbulent) Dissipation .................................... 26
3.3.4 Mixing length .................................................. 27
3.3.5 Reynolds Time Averaging .................................. 27
3.3.6 Averaging the Navier-Stokes equations ................. 28
3.3.7 Closure problem .............................................. 28
3.3.8 $k$-equation .................................................. 28
3.3.9 $\epsilon$-equation ............................................ 29

3.4 Turbulence Modelling ............................................ 30
3.4.1 One-equation model ......................................... 30
3.4.2 Two-equation models ....................................... 31
  3.4.2.1 The $k$-$\epsilon$ model ................................ 31
  3.4.2.2 The $k$-$\omega$ model ................................ 31
  3.4.2.3 The $k$-$\omega$SST model ................................. 31

3.5 Turbulent Boundary Layers .................................... 31
  3.5.1 Roughness .................................................. 33

3.6 Boundary Conditions ............................................ 35
  3.6.1 Inlet ....................................................... 35
  3.6.2 Surface boundary conditions ............................. 35
  3.6.3 Surface Roughness ........................................ 36

3.7 Mixture flows .................................................... 36
  3.7.1 Viscosity of slurry ....................................... 36
  3.7.2 Drift flux model .......................................... 36

3.8 Computational Fluid Dynamics ............................... 36
  3.8.1 Algorithms ................................................ 37
    3.8.1.1 SIMPLE algorithm .................................. 37
    3.8.1.2 PISO algorithm ..................................... 38
  3.8.2 OpenFOAM ................................................ 38
    3.8.2.1 Solvers .............................................. 38
      driftFluxFoam solver / algorithm ....................... 38
    3.8.2.2 Boundary conditions ................................ 38
      kqRWallFunction .......................................... 39
      kLowReWallFunction ................................. 39
      epsilonWallFunction .................................. 39
      epsilonLowReWallFunction .......................... 40
      omegaWallFunction .................................... 40
      nutUSpaldingWallFunction ......................... 41
      nutkRoughWallFunction .................. 41
      nutURoughWallFunction .......................... 42
## CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.8.2.3 Inlet</td>
<td>42</td>
</tr>
<tr>
<td>turbulentMixingLengthDissipationRateInlet</td>
<td>42</td>
</tr>
<tr>
<td>turbulentIntensityKineticEnergyInlet</td>
<td>42</td>
</tr>
<tr>
<td>4 Validation OpenFOAM</td>
<td>43</td>
</tr>
<tr>
<td>4.1 Introduction</td>
<td>43</td>
</tr>
<tr>
<td>4.2 General</td>
<td>44</td>
</tr>
<tr>
<td>4.2.1 Boundary conditions</td>
<td>46</td>
</tr>
<tr>
<td>4.3 Validation Cases</td>
<td>48</td>
</tr>
<tr>
<td>4.3.1 Eriksson</td>
<td>48</td>
</tr>
<tr>
<td>4.3.1.1 Simulation</td>
<td>48</td>
</tr>
<tr>
<td>4.3.1.2 Results</td>
<td>49</td>
</tr>
<tr>
<td>4.3.2 Rostamy</td>
<td>54</td>
</tr>
<tr>
<td>4.3.2.1 Simulation</td>
<td>55</td>
</tr>
<tr>
<td>4.3.2.2 Results</td>
<td>55</td>
</tr>
<tr>
<td>4.3.3 Förthmann</td>
<td>59</td>
</tr>
<tr>
<td>4.3.3.1 Simulation</td>
<td>59</td>
</tr>
<tr>
<td>4.3.3.2 Results</td>
<td>60</td>
</tr>
<tr>
<td>4.4 Conclusion</td>
<td>60</td>
</tr>
<tr>
<td>5 Simulating the flow in the scour hole</td>
<td>63</td>
</tr>
<tr>
<td>5.1 Introduction</td>
<td>63</td>
</tr>
<tr>
<td>5.1.1 Model Decisions</td>
<td>63</td>
</tr>
<tr>
<td>5.1.2 Boundary Conditions</td>
<td>63</td>
</tr>
<tr>
<td>5.2 BAGT-540</td>
<td>66</td>
</tr>
<tr>
<td>5.2.1 Hydraulically Smooth Boundary</td>
<td>66</td>
</tr>
<tr>
<td>5.2.2 Transitional Rough Boundary</td>
<td>66</td>
</tr>
<tr>
<td>5.2.3 Hydraulically Rough Boundary</td>
<td>66</td>
</tr>
<tr>
<td>5.2.4 Kinematic boundary</td>
<td>69</td>
</tr>
<tr>
<td>5.2.5 Roughness parameter E</td>
<td>69</td>
</tr>
<tr>
<td>5.3 SSB</td>
<td>70</td>
</tr>
<tr>
<td>5.4 Evaluating the jet model results</td>
<td>70</td>
</tr>
<tr>
<td>5.4.1 Velocities</td>
<td>70</td>
</tr>
<tr>
<td>5.4.2 Erosion</td>
<td>72</td>
</tr>
<tr>
<td>5.4.3 Shear velocity</td>
<td>72</td>
</tr>
<tr>
<td>5.5 Conclusions</td>
<td>75</td>
</tr>
<tr>
<td>6 Conclusions and Recommendations</td>
<td>77</td>
</tr>
<tr>
<td>6.1 Conclusions</td>
<td>77</td>
</tr>
<tr>
<td>6.2 Recommendations</td>
<td>77</td>
</tr>
</tbody>
</table>
# CONTENTS

A Two-equation models
  A.1 The $k$-$\epsilon$ model ............................................. 83
  A.2 The $k$-$\omega$ model ............................................. 84
  A.3 The $k$-$\omega$SST model ........................................... 84

B Drift flux model ....................................................... 87
  B.1 Mixture ............................................................... 87

C Plane Wall Jet .......................................................... 91
  C.0.1 Equations .......................................................... 91
  C.0.2 Found relations ................................................... 92

D Plane Wall Jet Grid Generation ....................................... 97

E Open Channel
  E.1 Introduction ......................................................... 99
  E.2 Simulation ............................................................ 99
  E.3 Conclusion ........................................................... 99

F Approximation of shear stress influence ........................... 103
  F.1 In pipe ............................................................... 103

G Permeability .................................................................. 105

H Chronologic overview .................................................... 107
List of Figures

1.1 The TSHD Vox Máxima (Van Oord) sailing ........................................... 1
1.2 Schematic drawing of a draghead, suction is applied to transport the mixture to the water surface, ripping and jetting will cause the sand bed to be fluidized ......................................................... 3

2.1 2D Sketch of velocity direction of traversing impinging jet. The grey part here is the nozzle the light blue is the ambient water and the darker blue is the jet flow. The sand bed is coloured yellow. ............................ 6
2.2 Screen capture from Sketchup model viewing a extended jet boundary (red), the sand bed is created (yellow) and the nozzle tube is positioned above (blue) the inlet is hidden behind the tube but is situated at the bottom of the tube ................................................................. 9
2.3 Sketch of a plane (or 2D) turbulent wall jet. ......................................... 11
2.4 Velocity profiles for different positions in the direction of the stream (x-direction), digitalised from (Förthmann, 1934). .................................................. 12
2.5 Similarity of velocity profiles calculated from digitalised data from (Förthmann, 1934). ................................................................. 13
2.6 Variation of the velocity scale for plane jets. The yellow collected by (Rajaratnam, 1976), the red collected by (George et al., 2000) and the blue collected by (Launder and Rodi, 1981). ......................................... 14
2.7 The velocity in the turbulent boundary layer region, non-dimensionalised for different nozzle Reynolds numbers. Data digitalised from different studies. The two profiles near the outlet (x/b₀ = 0 and 5) differ from the similarity probably because here the turbulent layer is not yet developed. Note that the expansion of the wall and the different slot heights are resulting in the varying turning points. ................................................. 15
2.8 Velocity scales for plane wall jets digitised from (Rajaratnam, 1976), added Eq. 2.11 ........................................................................ 16
2.9 The skin friction coefficient versus the local Reynolds number, experimental results from Erikkson and Abrahamson together with rough data found in the report of Rostamy see Section 4.3.2. Also the empirical found relation from Sigalla Eq. 2.14 and Eriksson Eq. 4.2. ................................. 17
LIST OF FIGURES

2.10 Velocity scales for the three-dimensional wall jets (Agelin-Chaab and Tachie, 2011), and Eq. 2.17. .............................. 18
2.11 Three dimensional wall jet .................................................. 19
2.12 Van Rijn’s digitized curves showing states of sand / water. ................. 21

3.1 Typical non-dimensional near wall velocity profile .......................... 33

4.1 Dimensions of all plane wall jet simulations. $L$ and $H$ are the dimensions as shown in Figure 4.1 ................................. 44
4.2 A close up of a typical grid layout, also shown is the near wall region where a higher refinement is chosen as the far wall region. .......... 45
4.3 Comparison of maximum velocity decay for the Eriksson case: OpenFOAM simulation results are shown for different grid sizes and wall functions, (a) are calculated using the k-$\epsilon$ and (b) the k-$\omega$SST turbulence model. Also shown is the maximum velocity decay equation Eq. 2.7 fitted on the experimental data from (Eriksson et al., 1998) by $C_1 = 3.55.$ .......................... 50
4.4 Comparison of the scale growth (b) for the Eriksson case: OpenFOAM simulation results are shown for different grid sizes and wall functions, (a) are calculated using the k-$\epsilon$ and (b) the k-$\omega$SST turbulence model. Also shown is the experimental data from (Eriksson et al., 1998). ......................... 51
4.5 Friction coefficient $c_f$ versus the local Reynolds Number $Re_m$ for (a) k-$\epsilon$ and (b) k-$\omega$SST model. Also shown are experimental results from Eriksson and Abrahamson taken from (Eriksson et al., 1998), together with Sigalla’s relation shown in Eq. 2.14 ........................................ 52
4.6 The near wall velocity profiles calculated from the simulation results for the Eriksson case with (a) k-$\epsilon$ and (b) k-$\omega$SST turbulence model on grid B and the high Reynolds wall functions shown as the continuous lines. Together with the Eriksson experiment and the Spaldings law of the wall function. ................................................ 53
4.7 The inner velocity scale of the smooth (Eriksson) and rough (Rostamy) turbulent wall jet, nondimensionalised for different nozzle Reynolds numbers. Data digitalised from (Rostamy et al., 2010) and (Eriksson et al., 1998). The log wall function Eq. 3.38b and Spaldings’ wall function Eq. 3.40 are also plotted for the calculated $B$ and $E$ using Eq. 3.45 for $k_s^+ \approx 25$ and $\kappa$ set to the standard value $\kappa = 0.41.$ ........................................ 54
4.8 The near wall velocity profiles calculated from the simulation results for the Rostamy case with (a) k-$\epsilon$ and (b) k-$\omega$SST turbulence model on grid B and the high Reynolds wall functions shown as the continuous lines. Together with the Rostamy experiment and the Spaldings law of the wall function (Eq. 3.40) for $E = 1.063$ and $\kappa = 0.41.$ .............................. 56
4.9 Friction coefficient $c_f$ versus the local Reynolds Number $Re_m$ for k-$\epsilon$ and k-$\omega$SST model. Also shown are experimental results from Rostamy and Tachie taken from (Rostamy et al., 2010), together with Sigalla’s relation shown in Eq. 2.14 ........................................ 57
4.10 Comparison of maximum velocity decay for the Rostamy case: OpenFOAM simulation results are shown for different grid sizes and wall functions, calculated using the k-\(\varepsilon\) and the k-\(\omega\)SST turbulence model. Also shown is the maximum velocity decay curve for \(C_1 = 3.2\) this coefficient is fitted on the simulation data for the k-\(\omega\)SST model.

4.11 Velocity scales for plane wall jets calculated form (Förthmann, 1934), together with the maximum velocity decay Equation 2.7 for \(C_1 = 4.55\) and the modified Equation 4.3 for \(C_1 = 3.20\).

4.12 Friction coefficient \(c_f\) versus the local Reynolds Number \(Re_m\) for k-\(\varepsilon\) and k-\(\omega\)SST model for the Förthmann case together with Sigalla’s relation shown in Eq. 2.14.

4.13 Results of the simulation of the Förthmann case 4.13 shows the maximum velocity decay, it shows that the k-\(\omega\)SST model gives the best results. Note that the k-\(\varepsilon\) models shows an early wall separation of the flow, therefore the curve stop at a shorter distance from the inlet.

5.1 Face names of the jet model, also the dimensions are shown for length \(L\), width \(W\) and height \(H\).

5.2 Results of simulations with roughness parameters (see the turbulent boundary layer discussion in Section 3.5) \(E = 9.8\), \(E = 1.063\) and \(E = 0.266\).

5.3 Chords, blue the bottom chord and red the side chord, where the bottom is defined as the deepest point and side as the widest point for each position in \(x\)-direction.

5.4 Simulation results kinematic erosion velocity plotted versus the calculated erosion velocity (a) BAGT-540, (b) SSB set 1 and (c) SSB set 2.

5.5 Simulation results shear velocity plotted versus the calculated (a) BAGT, (b) SSB set 1 and (c) SSB set 2.

C.1 Velocity scales for plane wall jets digitised from (Rajaratnam, 1976), added Eq. C.6 for \(C_1 = 3.65\), the blended function Eq. C.11 with \(a_2 = 9.734\) and \(p_2 = 6\) and a limited function Eq. C.10.

C.2 The skin friction coefficient versus the local Reynolds number, experimental results from Erikkson and Abrahamson together with rough data found in the report of Rostamy see Section 4.3.2. Also the empirical found relation from Sigalla Eq. C.15 and Eriksson Eq. 4.2.

C.3 Variation of the velocity-decay coefficient with relative roughness digitised from (Rajaratnam, 1976) and a curve fitted relation Eq. C.20.

C.4 Roughness effects on the velocity scales for plane rough wall jets digitised from (Rajaratnam, 1976), added Eq. C.6 for \(C_1 = 3.65\) representing the smooth wall, and Eq. C.18 representing rough wall with roughness \(k_e/b_0\).

E.1 Results of the open channel simulation at \(x = 8m\), two turbulence models are compared with Coleman experiments.
F.1 Principal drawing used to estimate the extra shear stress due to mass injection in a pipe . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 104

H.1 Summary of project outline . . . . . . . . . . . . . . . . . . . . . . . . . . 108
List of Tables

4.1 Grid specifications used for simulating the plane wall jet, for the near wall region and far from region the number of cells in x,y,z direction are given, with the expansion factor \( R \) for the grid grading towards the bottom marked with blue as explained in Appendix D. 45

4.2 The boundary conditions used for all plane wall jet models using both the \( k-\epsilon \) or \( k-\omega SST \) turbulence model. 46

4.3 The boundary conditions applying for the plane wall jet models using the \( k-\epsilon \) or \( k-\omega SST \) turbulence model and low and high Reynolds wall functions. 47

5.1 The boundary conditions applying for all jet models using the \( k-\omega SST \) turbulence model. 65

E.1 The boundary conditions applying for the open channel simulation using the \( k-\epsilon \) or \( k-\omega SST \) turbulence model. 100
Chapter 1

Introduction

1.1 General problem

Dredging is the activity of excavation, transportation and deposition of soil (clay, silt and sand) and/or rock in a wet environment. This is done for example as deepening of ports and waterways or for the creation of artificial islands.

Figure 1.1: The TSHD Vox Máxima (Van Oord) sailing
The equipment used is called a dredger and can be divided into several types. Typical dredger used for maintenance of water ways is the so called Trailing Suction Hopper Dredger (TSHD), like seen in Figure 1.1. This type of dredger removes soil hydraulically by trailing above and applying suction to the (sand) bed. Hydraulic dredging is based on eroding the sand bed by a near bed flow resulting in a sand and water mixture which enters the suction pipe. Dredge pumps provide the suction needed to transport the mixture to the surface and store it generally inside the hopper. The typical trailing speed is 0.5 to 1.5 m/s and mixture flows of 40 to 100 m³/min are reached. Maximum depths are in typical 100 m. The equipment where this happens is called the draghead and is shown schematically in Figure 1.2.

In a commercial operating company production and efficiency are one of the most important aspects. The production is often defined as the quantity of soil displaced per unit of time and one can guess that the optimal production is closely related to the efficiency.

In the 80’s the jetting of sand at the draghead was introduced. For trailer suction hoppers, jetting of sand is an important part of the dredging process. The production of the sand is done by a very large part of the jetting process, therefore a lot of time and effort is given to predict the process. Estimations of productions were done by empirical correlated models and later on a model, based on mostly physical aspects, was introduced. The latter can not only predict the penetration depth and average jet body width but also its shape due to the propagation of the nozzle.

In order to gain more insight in the relevant processes within the jet process, experiments by SSB were done at Deltares. These experiments made it possible to provide some validation of (and updates for) the existing models. Using conductivity porosity measurements concentrations of sand in the scour hole, which is the hole formed because sand is eroded, could be determined on fixed points in the sand bed by using a correlation. The models predicted less penetration depth and a larger width than observed in the mentioned measurements. In Figure 2.1 a schematic drawing of model and experiments is shown. This is the erosion process shown in Figure 1.2. The nozzle is oriented so that the flow at the outlet is directed perpendicular to the (in situ) sand bed. The translation of the nozzle and the erosion of the bed results in a so called scour hole typically shaped like shown in Figure 2.1.

The models generally use erosion relations based on pick-up (and sedimentation) fluxes. The pick-up flux can be calculated using the a pick-up function. The calculated pick-up depends on several bed properties, such as grain size diameter and porosity etc., and flow properties near the bed such as the flow induced shear stress. The shear stress on his turn is related to the (very) near bed velocity gradient and the near bed fluid viscosity. The earlier mentioned models use assumed friction factors also used for example in pipe flow.

Often the velocity of the flow is taken to be uniform along stream downwards. The latter assumption is rather rough. For the conducted experiments however no velocities were measured in the experiments (except for 1 case, but no near bed velocity was measured here). The reason these velocities are not measured is because of the difficulty
to create a set-up where these velocities can be measured. Firstly it is unclear where bed exactly is positioned, this makes it difficult to use rather simple measurement equipment such as pivot tubes and EMS-sensors as they are fixed on 1 point.

Even if these measurement techniques could be used one should keep in mind that the flow is most likely influenced by the present equipment. Secondly the concentrations of sediment in the flow will make it impossible to use laser based measurement techniques. These would, if possible to use in the same set-up as the experiments done, have no effect on the flow and the unknown bed location would most like introduce no problems (or using this technique can even be measured).

Despite the limits, discussed above, the scour hole can be used to determine the erosion velocities. This can be done assuming that the scour hole shape is stationary in time and if using a translating with the nozzle coordinate system also stationary in space. One can now determine the from here on so called kinematic erosion velocity, for example from the normal surface vectors of the scour hole and the translation velocity.

As mentioned above the uniform velocity assumption is questionable and not much reliable experimental data is available the question raises what would a better approximation be and what bed shear stress is applied by the flow. This project is mainly based on the attempt to use a numerical model for determining these unknown parameters.
CHAPTER 1. INTRODUCTION

Starting point is the measured scour hole data.

1.2 Goal

The lack of velocity data as discussed in the previous section, and the difficult circumstances for measuring the velocities, lead to the main goal of this thesis:

*Determine the flow velocities in the measured scour hole.*

Additional objectives are formulated:

*Find a simple relation to determine the velocities in the scour hole.*

and

*Compare the kinematic erosion velocities with the ones calculated using the velocities obtained in the main goal.*

1.3 Approach

To reach the goal the work presented can be divided into different parts: a literature study, simulations and finally an evaluation. These parts also have a division because there is limited amount of validation data for the simulations of plane wall jets which are done first, this should give some insight in the tools used for simulating the jet flow in the scour holes.

The Literature study is given in the chapters 2 and 3. Chapter 2 starts with the traversing jet, then the plane wall jet is discussed and finally something is told about erosion process. The third chapter discusses theory used for flow description for mainly single phase flow. A very short notion is given about the multiphase or mixture flow and sand bed interaction (i.e. erosion) is also discussed. Chapter 3 also discusses the methods used to solve the flow numerically (in other words: using CFD).

Chapter 4 gives a description of the numerical solution done to solve the plane wall jet problems, at the end of this chapter the results are discussed. The discussion gives insight in what to expect and which solvers can be used in the simulations of flow in the scour hole. The set-up of these jet flow simulations are discussed in chapter 5, at the beginning a short descriptions of a calibration (by choosing the roughness parameter) is done on BAGT data. Then a selected set of experiments form SSB are simulated. The model results are given and discussed and the overall conclusions and recommendations are given in the last chapter.
Chapter 2

Traversing impinging jet on sand

2.1 Introduction

In attempting to increase the insight in all the processes in a draghead like shown in Figure 1.2 one needs to zoom in and consider each process individual. The jet flow erosion is considered here, like shown in Figure 2.1. Several studies in eroding translating jets are present like (Weegenaar, 2011), however no studies are found where one discusses the velocity distributions itself. The van Rhee model for example assumes an uniform velocity profile along the stream downward direction, and for example friction is related to this assumed velocity profile. Also some experiments were conducted, from which shown in Figure ?? is a frame still of a high speed camera film of a jet experiment. The flow is visualised using some colour, showing the basis of our model.

The flow of water causes erosion and probably also a “mechanical” excavation resulting in a scour hole. Water entrainment from the surroundings, ambient and pore water as well as sand particles, is present. The jet nozzle moves over the sand bed and the jet flow direction is opposite of this the traversing jet nozzle see Figure 2.1. Further more an excavation process is observed similar to the breaching process (moving wall) at the side walls. When looking sufficiently far away the flow velocity is eventually low enough to let (partially) settle the sand particles again on the bottom, however this will occur at distances so far away from the inlet where no practical interests are left.
Figure 2.1: 2D Sketch of velocity direction of traversing impinging jet. The grey part here is the nozzle the light blue is the ambient water and the darker blue is the jet flow. The sand bed is coloured yellow.
2.1.1 Van Rhee Model
Deleted

2.1.2 BAGT-540
Deleted
2.1.3 SSB-P19

Deleted

2.1.4 Creating scour hole surfaces

Using the lines shown in Figure ?? one can construct a 3D model by creating a surface through each of the contour lines. Assuming symmetry along the plane \( y = 0 \) one can obtain a representation of the body similar as shown in Figure ?? . The symmetry will become in handy later when the model is calculated: it will reduce calculation time and memory requirements significantly. Note that for the SSB experiments this step is not necessary because a volume model is already present for each measurement.

The BAGT experiment STL Files were created using Autodesk’s AutoCAD 3D\(^1\): the ‘estimated’ scour holes given in Figures ?? and ?? were digitized and imported, and then the surfaces between these slices were created.

For the SSB experiment this step was not needed because the scour hole was digital already available from analysis (van der Hout, 2010). In order to create the correct domain (e.g. the sand bed, nozzle tube and the inlet) the scour hole was modified using Trimble’s Sketchup\(^2\). A screen shot of one of the models created using this method is shown in Figure 2.2.

The 3D model is then used to create the flow domain, and the surfaces such as outlet, top and the surrounding other are ‘automatically’ created by the intersection of the STL model and the uniform mesh created with the blockMesh utility. Despite that the scour hole is assumed to be stationary in time and place, the method used to solve the flow field is in-stationary in time. The reason is mainly because of the method, chosen at first and never changed, makes it easier to create a stable solution.

Looking at Figure 2.2 one can see that the boundary determined is not resulting in a ‘wall’ jet because of the offset. The boundary assumed is therefore questionable.

2.1.5 ‘Bottom’ and ‘Side’

We also define two points of interest for each position in x-direction: The point at the bottom, this is where the \( z \)-coordinate is maximal (absolute) and most of the time this points is situated at \( y = 0 \). These points are called from here on as the ‘bottom’ points:

\[
P_{\text{bottom}} = P(x, y, z) \quad \text{where} \quad z(x) = z_{\text{max}}(x)
\]

The second point of interest is the point where the the \( y \) coordinate (absolute) is maximum for an arbitrary \( x \), called from here on as the ‘side’:

\[
P_{\text{side}} = P(x, y, z) \quad \text{where} \quad y(x) = y_{\text{max}}(x)
\]

\(^1\)AutoCAD 3D is a broadly used computer aided drawing program
\(^2\)Sketchup is a computer program which is used for fast and easy three dimensional models
2.1.6 Erosion velocities (kinematic)

Now if one knows the (outward facing) surface normal $\mathbf{n}_{\text{surf}}$, one can easily calculate the erosion on each point of the body if it is assumed that the shape will be constant. If so, then by using $\mathbf{v}_t$ which is the trailing speed of the jet nozzle ($[0.75; 0; 0]\text{m/s}$ for the considered experiment of BAGT) one can determine the erosion speed by projecting the trailing speed vector on the surface normal vector:

$$v_e = \mathbf{n}_{\text{surf}} (\mathbf{n}_{\text{surf}} \cdot \mathbf{v}_t) \quad (2.3)$$

The notation $\cdot$ represents the inner product of the two vectors.

If only looking at $P_{\text{bottom}}$ and $P_{\text{side}}$ the normal vector respectively lays in the $xz$-plane and $xy$-plane. The problem reduces from 3 to 2 dimensions, as if the surface tangent here is known one can create a (normalized) tangent vector finally by rotating this (with an angle of $-\pi/2$) to obtain a normal vector.

Also known is the tangent of the curve at this point is equal to its first derivative at that point. So for example an vector in $xy$-plane where one knows its derivative $f' = dy/dz$ a vector parallel to the tangent of the point passing through $(0, 0)$ is:

$$\mathbf{v}_{\text{tangent}} = \begin{bmatrix} 1 \\ f'(1) \end{bmatrix} \quad (2.4)$$

$$\mathbf{n}_{\text{surf}} = \frac{\mathbf{v}_{\text{tangent}}}{||\mathbf{v}_{\text{tangent}}||} \begin{bmatrix} \cos(-\pi/2) & -\sin(-\pi/2) \\ \sin(-\pi/2) & -\cos(-\pi/2) \end{bmatrix} \quad (2.5)$$

The notation $||...||$ represents the length of the vector surrounded by this. Note that the...
length of a vector stays the same after the multiplication with the rotation matrix is done.

As a result of the linear interpolation, used to create the 3D model, the derivatives calculated by discrete approach is somewhat unsatisfactory. This could have been avoided using splines to create the intermediate surfaces, however this would have taken to much effort. Instead a continuous curve is fitted through the points, as can be seen in Figure ??, the (continuous) curve and its derivative together with several normal vectors are also shown in these figures.

Now the erosion velocities for side and bottom can be calculated using Eq. 2.3, the results are shown in Figure ??.

Deleted
2.2 Wall Jet

The experiments conducted earlier resulted in the scour hole, or jet body. In many of these experiments it was observed\(^3\) that a plane wall jet like flow was present. This is the reason that the existing plane wall jet theory is investigated here. Many different types of jets are discussed in literature such as wall jets, but also impinging jets and free jets are thoroughly investigated.

![Figure 2.3: Sketch of a plane (or 2D) turbulent wall jet.](image)

The plane (or 2D) turbulent wall jet is discussed in textbooks such as (Rajaratnam, 1976), the basis of which the theory/model is build upon are experiments done by ( Förthmann, 1934), a review of many studies is done by (Lauder and Rodi, 1981). Another review and collection of studies is done by (George et al., 2000).

Most (if not all) experimental work is done using air as medium, however the experiments by (Eriksson et al., 1998) is conducted using water, similarity between the air and water experiments can be accomplished by maintaining the same (nozzle) Reynolds number. The last study the turbulent velocity fluctuations are measured and is very well documented. A basic sketch of some parameter definitions is given in Figure 2.3. The parameters describing the situation are:

- \(b_0\): the slot height in m.
- \(b\): the vertical location where the velocity is half the maximum velocity, for a certain \(x\) in m.
- \(\delta\): the vertical location where the velocity is maximum, for a certain \(x\) in m.
- \(U_0\): the initial velocity in m/s.
- \(U_m\): the maximum velocity for a certain \(x\) in m/s.
- \(x_0\): the transition point from development region to developed region in m.

\(^3\)See for example (?)
Due to wall-fluid interaction the velocity profile will continuously vary along horizontal axis, assumed that it starts as a uniform profile with a very thin boundary layer. This boundary layer develops along the wall. The jet also has a so-called penetrating shear layer, here the entrained water enters the jet flow region. Typically the flow is divided into two regions:

- **Development region**: this is the region where the maximum horizontal velocity $U_m$ equals $U_0$.
- **Developed region**: this is the region where $U_m < U_0$.

### 2.2.1 Plane Wall Jet Similarity

Fürthmann concluded that for the plane wall jet the velocity profiles tend to a generalised form, the data from his report is given in Figure 2.4, here the data is digitised. The profiles tend to a generalised form which is easily seen when plotting $U/U_m$ over $\eta = z/b$, as shown in Figure 2.5. Here the first few positions are not confirming this profile because of the inlet velocity is still the maximum velocity which indicate that the jet is not yet developed.
2.2.2 Equations

The similarity equations discussed more in Appendix D, the results are given now. From (Rajaratnam, 1976) we know that:

\[ U_m \propto \frac{1}{\sqrt{x}}, \quad b \propto x, \quad \tau_w \propto \frac{1}{x} \]  
(2.6)

Further using dimensional considerations one gets:

\[ \frac{U_m}{U_0} = C_1 \sqrt{\frac{x}{b_0}} \]  
(2.7)

\[ b = C_2 x \]  
(2.8)

\[ \tau_w = C_3 \frac{\rho U_0^2}{2(x/b_0)} \]  
(2.9)

The coefficients \( C_1, C_2 \) and \( C_3 \) are to be determined.

2.2.3 Experimental results

The coefficient \( C_1 \) can be determined using the relations defined earlier and experimental results, Rajaratnam used data of Myers. However the collected work in (Launder and Rodi, 1981), which are several studies at different nozzle Reynolds numbers, suggest another coefficient. Also (George et al., 2000) used different experimental data and also his plot suggests another coefficient. When looking at Figure 2.6 we see that these
studies vary much and that there might be a dependence on the nozzle Reynolds number when looking at each individual collected data.

One should note that Eq. 2.7 implies that the maximum velocity at \( x/b_0 = 0 \) would be infinitely large, which in fact is not true because it would be maximal 1 because the maximum velocity is here the inlet velocity.

The velocity scale in the boundary layer is given in Figure 2.7 showing digitalised data from different studies like Irwin (Irwin, 1973) (which is performed on a positive pressure gradient) and Karlsson where the data is taken from (Gerodimos and So, 1997) and Eriksson taken from (Eriksson et al., 1998) and (George et al., 2000).

2.2.4 Found relations

The similarity of the wall jet is determined by Verhoff and can be found in (Rajaratnam, 1976)

\[
\frac{U}{U_m} = 1.48\eta^{1/7}[1 - \text{erf}(0.68\eta)]
\]  

(2.10)

in this \( \eta \) is the non dimensional height defined as \( \eta = z/b \) and erf is the so called error function.

The coefficient \( C_1 \) is taken in (Rajaratnam, 1976) as 3.50 and invariant of the nozzle Reynolds number. However when trying to fit Eq. 2.7 on the data shown in Figure 2.6 one might as well choose \( C_1 \) equal to 3.65, note that these lines are not shown here. If
Figure 2.7: The velocity in the turbulent boundary layer region, non-dimensionalised for different nozzle Reynolds numbers. Data digitalised from different studies. The two profiles near the outlet ($x/b_0 = 0$ and 5) differ from the similarity probably because here the turbulent layer is not yet developed. Note that the expansion of the wall and the different slot heights are resulting in the varying turning points.
using the second given value and the fact that the maximum velocity \( u_m \) any where can not exceed the inlet velocity \( U_0 \) (if the profile is assumed to be developed) the velocity scale of the jet can be described by:

\[
\frac{U_m}{U_0} = \min \left( 1; \frac{3.65}{\sqrt{x/b_0}} \right) \quad \text{for} \quad \frac{x}{b_0} \leq 100
\]

Equation 2.11 is shown together with data found in (Rajaratnam, 1976) in Figure 2.8.

The constant \( C_2 \) in Eq. C.7 used for velocity scale growth relation is determined to be equal to 0.068 so Eq.C.7 becomes:

\[
b = 0.068x
\]

The wall shear stress is rewritten, see for example (Rajararadnam, 1976), in two different forms and the empirically found relations also given. The results are two different skin friction factors, Myers reported:

\[
f \equiv \frac{\tau_w}{\rho U_0^2/2} \approx \frac{0.20}{x/b_0(U_0b_0/\nu)^{1/12}} = \frac{0.20}{x/b_0(Re_0)^{1/12}} = 0.20 \left( \frac{x}{b_0} \right)^{1/12}
\]

and Sigalla:

\[
f' \equiv \frac{\tau_w}{\rho U_m^2/2} = \frac{0.0565}{(U_m y_m/\nu)^{1/4}} = \frac{0.0565}{(Re_m)^{1/4}}
\]

Note in 2.13 the nozzle Reynolds number is present. In Eq. 2.14 the \( y_m \) represents the distance from the point where the velocity in the jet is maximal to the (nearest) surface as defined in Figure 2.3, most literature calls this the local Reynolds number defined as:

\[
Re_m = U_m y_m/\nu
\]
Figure 2.9: The skin friction coefficient versus the local Reynolds number, experimental results from Eriksson and Abrahamson together with rough data found in the report of Rostamy see Section 4.3.2. Also the empirical found relation from Sigalla Eq. 2.14 and Eriksson Eq. 4.2.

Reliable data on skin friction is reported by Eriksson (see Section 4.3.1) and is plotted together with Sigalla’s relation in Figure 2.9.

The entrainment velocity is expressed in (Rajaratnam, 1976) as:

\[ v_e = \frac{dQ}{dx} = \alpha_e U_m = 0.035 U_m \]  

(2.16)

The coefficient \( \alpha_e = 0.035 \) is the entrainment coefficient.

2.2.5 Experimental work

Further on in this report a brief description of experimental work found in literature are discussed, these cases are used to validate the plane wall jet simulations. Some data is already mentioned in this chapter but for the sake of clarity it is given again.

2.2.6 Turbulent three-dimensional wall jet

Although the title of this chapter says other wise the three dimensional wall jet is also discussed here, later on it will become clear that available theories can be used to our benefit. In (Agelin-Chaab and Tachie, 2011) the characteristics of a three-dimensional wall jet is discussed. Used for non dimensional length is the area equivalent diameter \( d_e \),
which for a round inlet is just the diameter of the nozzle. A schematic drawing is shown in 2.11, here also the general found profiles are drawn. It also given in (Rajaratnam, 1976) that round and square inlets show the same similarity characteristics. Fitting a line through these points is done for using later on, the equation proposed is:

\[
\frac{U_m}{U_0} = a_3 + b_3 \left( \frac{x}{d_e} \right) + c_3 \left( \frac{x}{d_e} \right)^3
\]

(2.17)

where \(a_3 = 161.8\), \(b_3 = 1.78\) and \(c_3 = -0.0848\). Above equation and the digitised data from (Agelin-Chaab and Tachie, 2011) are shown in 2.10. In this report, the assumption is made that one can state that \(b_0 = d_e\) to compare relations with simulation results. It is stressed here that this is not based on any physical consideration or whatsoever. Also note that now 2 velocity scale growth parameters are present: \(b_y\) for the vertical profile and \(b_z\) for the horizontal profile.

In Figure 2.10 also the maximum velocity decay of a free 3 dimensional circular jet is plotted:

\[
\frac{U_m}{U_0} = C_1 \left( \frac{x}{d_e} \right)
\]

(2.18)

In the plot is seen that only for the low nozzle Reynolds case the shape of the curve is conceding with the experimental results. It should be stressed that in the plot \(C_1 = 6\) is used (Rajaratnam, 1976) multiple studies are summed where the coefficient ranges from \(C_1 = 5.75\) to 7.32.

### 2.3 Erosion and Settling

An important process in the impinging jet is the erosion process and is discussed now.

---

4This is done without applying any dimensional analysis whatsoever.
Figure 2.11: Three dimensional wall jet
CHAPTER 2. TRAVERSING IMPINGING JET ON SAND

Different studies are available in literature where the erosion of a sand bed is modelled. Such studies give different approaches considering stability of the bed. For example Izbash approach (Shiereck, 2012) considers individual flow and grain interactions by addressing drag, shear and lift forces. Another approach is the method of Shields, which van Rhee used and modified and an simplification by Bisschop is available. The shields approach and its extended / modified models are used and discussed in this thesis. These approaches differ from the Izbash approach in a general way that not individual grains are considered but larger surfaces of sand bed and the shear stress acting on the bed.

2.3.1 Bed growth

The bed growth (or decrease) can be described using the definition (van Rhee, 2010):

\[ v_e = \frac{S^* - E^*}{\rho_s(1 - n_0 - c_b)} \tag{2.19} \]

The settling and erosion (or pick-up) flux are respectively denoted by \( S^* \) and \( E^* \). The density of sand particles \( \rho_s \), porosity of the (settled) bed \( n_0 \) and the near-bed concentration (volumetric) \( c_b \).

2.3.2 Settling and hindered settling

The settling flux can be expressed as:

\[ S^* = \rho_s w_s c_b \tag{2.20} \]

where \( w_s \) denotes the hindered settling velocity of the particles and its derivation can be found in several literature like (van Rhee, 2002) or (Bisschop et al., 2010). Because the settling of particles is influenced by the concentration of sand in the water.

\[ w_s = (1 - c_b)^n w_0 \tag{2.21} \]

The exponent \( n \) is a parameter which should be chosen for the corresponding Reynolds particle number, which can be found for example in (van Rhee, 2002) or in Appendix B. The terminal settling velocity \( w_0 \) for perfectly round particles can be determined by:

\[ w_0 = \sqrt{\frac{4g\Delta d}{3C_d}} \tag{2.22} \]

The parameter \( C_d \) is the drag coefficient of the particle depending on the Reynolds number defined as:

\[ Re_p = \frac{wd}{\nu} \tag{2.23} \]

The velocity \( w \) denotes the vertical velocity of the particle. The drag coefficient is to be chosen different for each regime: laminar, transitional and turbulent and can also be found in (van Rhee, 2002).
2.3.3 Pick-up

The pick-up flux can be determined by using the so called pick-up function, various functions are known using the Shields stability criteria $\theta_{cr}$, which is constructed considering the particles’ resistance to move over the bed versus the lift and drag force acting on the particles. In other words: for which $\theta$ flow does the bed start eroding, this results in the following expression:

$$\frac{u^2}{\Delta gd} = \theta$$  \hspace{1cm} (2.24)

Here $u_\tau$ is the friction velocity defined later in this text (see Eq 3.34), $\Delta$ denotes the specific density $\rho_s - \rho_w$, $g$ is the gravitational acceleration and $d$ the particle diameter. Originally Shields developed a diagram where the stability parameters where given (Shields, 1936), the diagram is modified by (van Rijn, 1984) and the last is digitised and shown in Figure 2.12. Also a curve which is fitted by (Brownlie, 1981) to data from Vanoni is available:5

![Figure 2.12: Van Rijn’s digitized curves showing states of sand / water.](image)

$$\theta_{cr} = 0.22Re_p^{0.6} + 0.06 \times 10^{-17.77}Re_p^{-0.6}$$  \hspace{1cm} (2.25)

The Reynolds number here is defined as the particle Reynolds number not using the settling velocity but:

$$Re_p^* = \frac{d\sqrt{\Delta gd}}{\nu}$$  \hspace{1cm} (2.26)

\hspace{1cm}

5Note that in the report of Brownlie an error in this relation is present!
At low velocities one can apply the pick-up function from (van Rijn, 1984):

\[ E^* = 0.00033 \rho_s \sqrt{g \Delta dD_0^{0.3}} \left( \frac{\theta - \theta_{cr}}{\theta_{cr}} \right)^{1.5} \]  

(2.27)

In this equation \( D_0 \) denotes the Bonneville parameter defined as (Miedema, 2012):

\[ D_0 = d_3 \sqrt{\frac{\Delta g}{\nu^2}} \]  

(2.28)

and one can easily see that \( D_0^{3/2} = Re_p^* \) by substituting Eq. 2.26 into Eq. 2.28.

In (van Rhee, 2010) it is shown that at high velocities the Shields’ stability criteria and the van Rijn equation (Eq. 2.27) overestimates the pick-up. To fix this extra terms accounting for entrainment due to dilatation resulting form bed shearing and the angle of inclination should be taken in to account:

\[ \theta_{cr}^* = \theta_{cr} \left( \sin \phi_b - \frac{\beta_b}{\sin \phi_b} + \frac{v_e}{\lambda_l} \cdot \frac{n_l - n_0 A}{1 - n_l A} \right) \]  

(2.29)

where \( \phi_b \) denotes the angle of internal friction of the sand, \( \beta_b \) the angle of inclination of the bed. The erosion velocity is denoted by \( v_e \) and \( \lambda_l \) denotes the permeability of the upper (dilated) layer. The porosities are denoted by \( n_l \) and \( n_0 \) which respectively is the porosity of the dilated layer and of the non-dilated layer (or in-situ porosity). \( A \) is a constant which is to be taken as \( A = 3/4 \) for single particle mode and \( A = 1/(1 - n_0) \) for continuum mode (\( A \approx 1.7 \)).

Now the erosion using this modified stability criteria can be expressed as:

\[ E^* = 0.00033 \rho_s \sqrt{g \Delta dD_0^{0.3}} \left( \frac{\theta - \theta_{cr}^*}{\theta_{cr}} \right)^{1.5} \]  

(2.30)

In (Bisschop et al., 2010) it is given that at high erosion velocities the inclination term can be omitted so that the critical shields parameter would be:

\[ \theta_{cr}^* = \theta_{cr} \left( \frac{v_e}{\lambda_l} \cdot \frac{n_l - n_0 A}{1 - n_l A} \right) \]  

(2.31)

Also shown in his report it is shown that the sedimentation flux \( S^* \) in Eq. 2.19 could be set to zero as the erosion process dominates.
Chapter 3

Computational Fluid Dynamics

3.1 Introduction

In the previous chapter it is discussed what processes are important in the translating jet. The erosion velocities are related to the shear velocity near the bed. In order to determine the shear velocity one should know the flow field of the jet. A method in doing so is for example creating a numerical solution: solve the flow numerically by discretization of space and time. This called computational fluid dynamics or CFD. In the next chapter the CFD code is validated on plane wall jet cases, with these results the next step is simulating the translating impinging jet which is discussed in the second chapter. A good start in understanding computational fluid dynamics is by looking at the basic flow equations. These are based on conservation laws and are discussed here briefly. Also the more complex equations for turbulence quantities and multiphase mixture are discussed here in some extent. After the basic equations and turbulence models are discussed the software package OpenFOAM is introduced, this is used in the next chapters to create a numerical solution.

3.2 Basic Equations

Most (if not all) textbooks concerning fundamental fluid mechanics like (White, 2011) give the derivations of the so called conservation laws. It basically implies taking a small control volume where inward and outward flows on all interfaces and, assuring that nothing can be created or destroyed in this volume, for the basic flow equations mass and momentum are taken as the relevant quantities.

3.2.1 Mass Conservation

The results of applying the conservation law for the quantity mass can be expressed in differential form:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x} (\rho u) + \frac{\partial}{\partial y} (\rho v) + \frac{\partial}{\partial z} (\rho w) = \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i)$$  (3.1)
Here $x, y$ and $z$ are the Cartesian coordinates which can be written compactly using the subscript notation $x_i$ where $i$ denotes each direction. And $u, v$ and $w$ denotes the fluid velocities in each direction $x, y$ and $z$ respectively. The density of the fluid is denoted by $\rho$. Another, rather compact, notation is using vectors:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (3.2)$$

Here $\mathbf{u}$ is the velocity vector, the gradient $\nabla (i \frac{\partial}{\partial x}, j \frac{\partial}{\partial y}, k \frac{\partial}{\partial z})$ and the inner product $\cdot$. Together denote the so called divergence, or $\text{div} = i \frac{\partial}{\partial x} + j \frac{\partial}{\partial y} + k \frac{\partial}{\partial z}$.

### 3.2.2 Linear Momentum

In the same textbooks earlier mentioned the derivation of the 3 linear momentum equations, one for each direction, can be found. Analogue to the derivation of the mass conservation law, these equations (one for each direction) are derived using the principle of conservation of momentum resulting in differential form:

$$\rho g_x - \frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} = \rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) \quad (3.3)$$

$$\rho g_y - \frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} = \rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) \quad (3.4)$$

$$\rho g_z - \frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} = \rho \left( \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) \quad (3.5)$$

Here $g_x$, $g_y$ and $g_z$ denotes the body forces in each direction, $p$ denotes the pressure and $\tau$ the stresses acting on the small block of fluid.

One can write this in a more compact notation as:

$$\rho \mathbf{g} - \nabla p + \nabla \cdot \mathbf{\tau} = \rho \frac{d\mathbf{u}}{dt} \quad (3.6)$$

where the upper-case straight $d$’s are used for the material derivative (or total derivative in general differential calculus where this would be denoted by lower case $d$’s):

$$\frac{d\mathbf{u}}{dt} = \frac{\partial \mathbf{u}}{\partial t} + u \frac{\partial \mathbf{u}}{\partial x} + v \frac{\partial \mathbf{u}}{\partial y} + w \frac{\partial \mathbf{u}}{\partial z} \quad (3.7)$$

and the shear stress tensor:

$$\mathbf{\tau} = \begin{bmatrix} \tau_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_{zz} \end{bmatrix} \quad (3.8)$$
### 3.2.3 Navier-Stokes Equations

When we apply Newtonian fluid properties to the Linear momentum equations we can derive the so called Navier-Stokes Equations. In a three dimensional incompressible Newtonian fluid the viscous shear stresses are $\mu$ linear proportional on the viscosity coefficient (White, 2011):

\[
\begin{align*}
\tau_{xx} &= 2\mu \frac{\partial u}{\partial x} \\
\tau_{yy} &= 2\mu \frac{\partial v}{\partial y} \\
\tau_{zz} &= 2\mu \frac{\partial w}{\partial z} \\
\tau_{xy} &= \tau_{yx} = \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \\
\tau_{xz} &= \tau_{zx} = \mu \left( \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right) \\
\tau_{yz} &= \tau_{zy} = \mu \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)
\end{align*}
\]

One can write the Navier-Stokes Equations as:

\[
\begin{align*}
\rho g_x - \frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) &= \rho \frac{du}{dt} \quad (3.9) \\
\rho g_y - \frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) &= \rho \frac{dv}{dt} \quad (3.10) \\
\rho g_z - \frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) &= \rho \frac{dw}{dt} \quad (3.11)
\end{align*}
\]

Note that the Navier-Stokes equations contain four unknowns $p, u, v$ and $w$. To find a solution the equations should be solved combined with the (incompressible) continuity equation (Eq. 3.1).

### 3.3 Turbulence

Most flows in nature are turbulent, which are observed as random eddies. Describing turbulence in a deterministic manner is therefore not possible. However we can still introduce some models to catch important turbulence properties. These models and quantities are discussed in this section briefly. First some definitions are given which are used, after that the methods used in this thesis for modelling the turbulence.
3.3.1 Turbulent energy

The turbulent energy is (Wilcox, 2010) defined as:

\[ k \equiv \frac{1}{2} u'_i u'_i = \frac{1}{2} \left( (u'_x)^2 + (u'_y)^2 + (u'_z)^2 \right) \]  (3.12)

The ’ denotes the fluctuating part of the velocity and is defined later this section by Eq. 3.21. It is good to note that Eq. 3.12 is energy per unit mass, and therefore sometimes called the specific turbulent kinetic energy.

3.3.2 Turbulence intensity

Turbulence intensity is defined as:

\[ I \equiv \frac{u_{rms}}{U_{mag}} \]  (3.13)

here \( u_{rms} \) represents the RMS of the turbulent fluctuations (often unknown) and \( U_{mag} \) represents the mean absolute velocity. Using the turbulent energy equation Eq. 3.12 the RMS value\(^1\) can be expressed Versteeg and Malalasekera (1995) as:

\[ u_{rms} \equiv \sqrt{\frac{2}{3} k} \]  (3.14)

and the mean velocity with

\[ U_{mag} \equiv \sqrt{U_i^2} = \sqrt{U_x^2 + U_y^2 + U_z^2} \]  (3.15)

Often Eq. 3.13 is used for estimating the turbulent inlet conditions, see also the section boundary conditions.

3.3.3 (Turbulent) Dissipation

The definition of the turbulent dissipation can be found in for example (Launder and Spalding, 1974) which states:

\[ \epsilon \equiv \nu \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_i}{\partial x_k} \]  (3.16)

Here \( \nu \) represents the kinematic viscosity of the fluid. The turbulent dissipation is still an unknown quantity in the \( k \)-equation, which will be introduced shortly, based models which are discussed later on in this chapter.

From (Wilcox, 2010) it is known (mainly based on dimensional analysis/considerations) that the dissipation of the eddies can are related by:

\[ \epsilon \propto \frac{k^{3/2}}{\ell} \]  (3.17)

\(^1\)RMS value of arbitrary variable \( \phi \) is defined as \( \phi_{rms} \equiv \sqrt{\frac{1}{n} (\phi'_1 + \phi'_2 + \ldots + \phi'_n)} \)
Prandtl proposes to use a dissipation constant $C_\mu$ resulting in:

$$\epsilon = C_\mu \frac{k^{3/2}}{\ell}$$

(3.18)

In this $C_\mu$ is a model constant and $\ell$ is the mixing length (see Section 3.3.4). Another definition of the dissipation could be the rate of destruction of the kinetic energy, mostly used definition is that of Kolmogrov and it can be found in (Wilcox, 2010, Celik, 1999):

$$\omega = c \frac{k^{1/2}}{\ell}$$

(3.19)

where $c$ is a constant, this definition is believed to be constructed mainly by dimensional analysis. It is easy to combine the found relations Eq. 3.18 and Eq. 3.19 resulting in:

$$\epsilon = \beta^* \omega k$$

(3.20)

where $\beta^*$ is also a constant and used later on in some two-equation models.

### 3.3.4 Mixing length

The mixing length $\ell$ is still an unknown. Normally the problem is solved (or often called ‘closed’) using simple algebraic equations to determine the eddy viscosity and a linear relation between $\ell$ and for instance the distance from an inlet, one can read for example a good introduction in (Versteeg and Malalasekera, 1995). However the mixing length approach is not used further in this thesis because the higher order turbulence models, which are discussed shortly in this chapter, are used.

### 3.3.5 Reynolds Time Averaging

One, rather simple, method for describing turbulent flows is using the so called Reynolds Averaging method. It basically means expressing any variable which can be depending on both time $t$ and space $x$ as a mean part and added a fluctuating part. So for an arbitrary variable $\phi(x,t)$:

$$\phi(x,t) = \Phi(x) + \phi'(x,t)$$

(3.21)

For the arbitrary quantity he mean value is thus $\Phi(x)$ and $\phi'(x,t)$ is the fluctuating part. By definition the average of the fluctuating component is zero. This results that for stationary turbulence, the mean part of the flow can be simplified by removing this fluctuating part:

$$\bar{\phi}(x,t) = \Phi(x) = \lim_{\Delta t \to \infty} \frac{1}{\Delta t} \int_{t}^{t+\Delta t} \phi(x,t) dt$$

(3.22)

Here $\Delta t$ should of be chosen much greater than the time scale of the turbulent fluctuations to get a good average. The over-line denotes taking the mean of these values.
3.3.6 Averaging the Navier-Stokes equations

If one applies the Reynolds time average method on the Navier-Stokes for mass and momentum the so called RANS (Reynolds Averaged Navier Stokes) equations are obtained:

\[
\frac{\partial U_i}{\partial x_i} = 0 \tag{3.23}
\]

\[
\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_i U_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} (T_{ji}) - \frac{\partial}{\partial x_j} (\rho u'_i u'_j) \tag{3.24}
\]

Equation 3.23 is the Reynolds averaged continuity equation, and Eq. 3.24 is the so called Reynolds averaged momentum equation. \(T_{ji}\) and \(P\) respectively denote the mean stress and pressure parts. Note the upper-case notation of the parameters represents the mean values, the lower-case and primed notation represents the fluctuating parts.

3.3.7 Closure problem

Because \(\rho u'_i u'_j\) (which is called the Reynolds stress tensor) is an unknown term in the newly found equations we need some extra equation(s) to model turbulence and to ’close’ the system because we have:

- 10 unknowns: the three velocity components, the pressure and the six stresses (\(\tau_{xy} = \tau_{yx}\))
- 4 equations: the continuity equation and the three components from the momentum equation

However as can be seen later on, new equations introduce even more unknowns. This so called closure problem is solved by introducing coefficients to close these extra equations which provide actually a model of turbulence. Several models are given in literature, in this paper three two-equation models are considered. These arise often from the (exact) turbulence kinetic energy equation and an energy dissipation equation which can be found in classical mathematical turbulence modelling discussed in the next sections.

3.3.8 \(k\)-equation

To obtain the turbulent kinetic energy equation one starts at the Navier-Stokes equation (for steady state, incompressible flow and constant viscosity) and subtracts the time averaged Navier-Stokes equation from that. Numerous manipulations (for example applying the definition of continuity equation) are then applied, which are omitted here and can be easily found by the reader in existing literature. The exact \(k\)-equation found is:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_j k)}{\partial x_j} = -\rho u'_i u'_j \frac{\partial U_i}{\partial x_j} - \rho \frac{\partial u'_j}{\partial x_j} \frac{\partial u'_i}{\partial x_j} - \frac{\partial}{\partial x_j} \left( \frac{1}{2} \rho u'_i u'_j \right) + \frac{\partial^2 (\mu k)}{\partial x^2_j} - \mu \frac{\partial u'_i}{\partial x_j} \frac{\partial u'_i}{\partial x_j} \tag{3.25}
\]
The terms in this (exact) equation for the turbulent kinetic energy are:

I Unsteady term
II Convection
III Production
IV Diffusion due to pressure and velocity fluctuations
V Diffusion due to velocity fluctuations (or Reynolds stress)
VI Diffusion due to viscous stress (or molecular diffusion)
VII Dissipation rate

The $k$-equation still has unknowns which appear in the terms III IV V and VII. These correlations can be modelled or one can introduce extra equations.

3.3.9 $\epsilon$-equation

Obtaining the $\epsilon$-equation starts from the (exact) definition, which is the equation of the dissipation given by Eq. 3.16. The mathematical form can be obtained by evaluating the following moment (Wilcox, 2010):

$$2\nu \frac{\partial u'_i}{\partial x_j} \frac{\partial}{\partial x_i} [N(u_i)] = 0 \quad (3.26)$$

The symbol $N(u_i)$ denotes the Navier-Stokes operator. The derivation of this equation can for example be found in (Celik, 1999), the result is:

$$\frac{\partial \epsilon}{\partial t} + \rho U_i \frac{\partial \epsilon}{\partial x_j} = -2\mu \left[ \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k} + \frac{\partial u'_k}{\partial x_i} \frac{\partial u'_j}{\partial x_m} \right] \frac{\partial U_i}{\partial x_j} - 2\mu \frac{\partial u'_i}{\partial x_j} \frac{\partial^2 U_i}{\partial x_k \partial x_k} \frac{\partial U_i}{\partial x_j} \frac{\partial^2 U_i}{\partial x_m \partial x_m} \quad (3.27)$$

Equation 3.27 shows many double and even triple correlations for the fluctuations, also it can be classified as a complicated equation. However (Wilcox, 2010, Celik, 1999) give some interpretations of the terms:

I Production of dissipation
II Dissipation of dissipation
III Destruction rate of dissipation due to the velocity fluctuations
IV Transport of dissipation due to viscous diffusion, velocity fluctuations and pressure-velocity fluctuations

And again unknown terms will remain which often are ‘closed’ using constants
3.4 Turbulence Modelling

Boussinesq (Davidson, 2011) approximates the stress tensor by linking the Reynolds stresses, arising from the RANS equation, to the velocity gradient (i.e. deformation) via the turbulent viscosity:

$$\tau_{ij} = -\rho u_i' u_j' = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)$$  \hspace{1cm} (3.28)

The turbulent dynamic viscosity $\mu_t$ is in units Pa·s and is assumed in the simple models to be constant for the considered flow, and later on, when discussing the two equation models, some expression for this are also given based on dimensional consideration. Note that the turbulent momentum transport is thus proportional to the mean velocity gradients.

Considering incompressible flow ($\frac{\partial U_i}{\partial x_j} = \frac{\partial U_j}{\partial x_i}$) and by using Eq. 3.12 and the continuity equation for steady state (see eq 2.9 and beyond in (Davidson, 2011)).

$$\tau_{ij} = \mu_t S_{ij} - \frac{2}{3} \rho \delta_{ij}$$  \hspace{1cm} (3.29)

The term $\delta_{ij}$ is the Kronecker delta, and $S_{ij}$ is the mean strain rate tensor i.e.:

$$S_{ij} = \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)$$  \hspace{1cm} (3.30)

The modelled Turbulence Kinetic Energy differential equation is:

$$\rho \frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \rho \epsilon + \frac{\partial}{\partial x_j} \left[ \left( \mu_t + \mu_t \sigma_k \right) \frac{\partial k}{\partial x_j} \right]$$  \hspace{1cm} (3.31)

Where the triple correlation in the exact $k$-equation is modelled using:

$$\frac{1}{2} \rho u_j' u_i' = \mu_t \frac{\partial k}{\partial x_j}$$  \hspace{1cm} (3.32)

Which is based on that the turbulent energy is diffused down the gradient. The closure coefficient $\sigma_k$ can be taken as a constant. Note that for a more complete derivation one should look into available literature.

3.4.1 One-equation model

One-equation models generally uses simple algebraic equations to model the turbulent viscosity, assuming that the characteristics of the turbulence in the flow are known ($\ell$). With a dissipation assumption, the $k$-equation can be closed using Eq. 3.17 resulting in the following one-equation model:

$$\rho \frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \rho C_p \frac{k^{3/2}}{\ell} + \frac{\partial}{\partial x_j} \left[ \left( \mu_t + \mu_t \sigma_k \right) \frac{\partial k}{\partial x_j} \right]$$  \hspace{1cm} (3.33)
3.4.2 Two-equation models

Two-equation models generally uses equations to model not only the turbulent viscosity but also the turbulence characteristics. Therefore a solution can be found without prior knowledge of the turbulence in the flow like needed in the one-equation models, also these models are complete. In this thesis three two-equation turbulence models are discussed and given in Appendix A in more detail, $k-\epsilon$, $k-\omega$ and $k-\omega_{SST}$ . The latter is actually a blend of the first two.

3.4.2.1 The $k-\epsilon$ model

The equation describing the turbulent kinetic energy is given by Eq. 3.31. To obtain a model $\epsilon$ equation (which describes the dissipation of the turbulent kinetic energy) the following terms of Eq. 3.27 are simplified the motivation can be found in (Celik, 1999) and results can be found in Appendix A in more detail.

3.4.2.2 The $k-\omega$ model

The rate of dissipation $\epsilon$ is not the only scale which turbulence is tried to be modelled with, looking at the turbulence frequency $\omega$ another model could be introduced. Kolmogrov\(^2\) (using $\omega \sim k^{1/2}/\ell$) introduced. Results can be found in Appendix A in more detail.

3.4.2.3 The $k-\omega_{SST}$ model

OpenFOAM also provides the turbulence model introduced by Menter: $k-\omega_{SST}$ where the last term stands for Shear Stress Transport, it provides a mix of the $k-\epsilon$ (any where outside the boundary layers, to avoid dependence of the free stream conditions) and the Wilcox $k-\omega$ model for inside the boundary layer (i.e. near the wall) (Menter, 1994). Menter formulated a so-called baseline model (BSL) and (Hellsten, 1997) provided an improvement introducing a blending for rough walls, the model is given in Appendix A in more detail.

3.5 Turbulent Boundary Layers

Near a (solid) wall the viscous effects on the turbulent flow become more important, this changes the behaviour of the fluid near these walls. When interaction of walls are important the fluid needs special attention in the non turbulent region.

Three types of wall models are discussed in (Sondak, 1994) the wall treatment used in this research is done with the so called wall function. The model arises from many experiments to determine the boundary layer of turbulent flow near walls, it was seen that different regions exist (see Figure 3.1):

\(^2\)Kolmogrov also stated that dissipation occurs at the smallest eddy scale thus the rate of dissipation is the rate of turbulent kinetic energy transfer from the larger to the smaller eddies
• **inner region** also called the law of the wall region, which can be subdivided into:
  – **viscous region**, where viscous effect dominates.
  – **transition region**, where the transition occurs from viscous dominated flow
to turbulence dominated flow (both dominate the flow).
  – **log region**, where turbulent stresses dominate the flow.

• **outer region** flow not affected by viscous effects but by inertial effects.

The law of the wall is build to be a general case, using different scales for relevant
parameters, and is discussed in this section. The velocity scale is defined as the near-
wall friction velocity:

\[ u_\tau \equiv \sqrt{\frac{\tau_{\text{wall}}}{\rho_{\text{wall}}}} \]  

(3.34)

The term \( \tau_{\text{wall}} \) denotes the wall shear stress and \( \rho_{\text{wall}} \) denotes the density of the fluid
at the wall. One can verify easily that the units of this indeed an velocity scale (m/s).

The wall shear stress can be found in several classical fluid mechanics textbooks:

\[ \tau_{\text{wall}} = \mu \left( \frac{\partial u}{\partial n_{\text{wall}}} + \frac{\partial v}{\partial n_{\text{wall}}} + \frac{\partial w}{\partial n_{\text{wall}}} \right) = \mu \left( \frac{\partial U}{\partial n_{\text{wall}}} \right) \]  

(3.35)

The component \( n_{\text{wall}} \) is the distance normal to the wall and the subscript \( \text{wall} \) indicate the
near wall partial derivatives which basically means \( n_{\text{wall}} = 0 \). The viscosity \( \mu \) here denotes
the viscosity. Also non-dimensional scales for velocity are used, defined by:

\[ u^+ \equiv \frac{U}{u_\tau} \]  

(3.36)

\[ y^+ \equiv \frac{u_\tau n_{\text{wall}}}{\nu} \]  

(3.37)

The velocity \( u \) here is the component tangent to the wall and the kinematic viscosity is
denoted by \( \nu \). The velocity profile can be described by:

**viscous region:** \[ u^+ = y^+ \]  

(3.38a)

**log region:** \[ u^+ = \frac{1}{\kappa} \ln \left( y^+ \right) + B \]  

(3.38b)

The term \( \kappa \) denotes the Karman constant which often is chosen in the range of \( 0.4 - 0.41 \) and
\( B \) is an empirically found constant in the range \( 4.9 - 5.5 \). These equations are plotted in Figure 3.1 where also the experimental found profile is shown.

In OpenFOAM a different notation is used representing the same relation as equation
\ref{3.38b}:

\[ u^+ = \frac{1}{\kappa} \ln \left( E y^+ \right) \]  

(3.39)

The constant \( E \) which similar like \( B \) is related to the roughness, a simple calculation
using the relation obtained from Eq. \ref{3.38b} and Eq. \ref{3.39}: \( B = \ln(E)/\kappa \) which results in
\( E = 7.0 - 9.8^3 \).

\footnote{Also Rajaratnam collected some expressions in (Rajaratnam, 1976) and noted that Mathieu Tailland
(1963) used \( \kappa = 0.25 \) and \( B = 10.30 \). Myers used \( \kappa = 0.18 \) and \( B = 4.9 \). These values (especially \( \kappa \)) lay
outside the domain which was given earlier!}
In Figure 3.1 the curve for a smooth boundary is shown, and a impression on how the curve would be for a rougher surface. In this Figure the $\Delta B$ is the difference for the parameter for rough walls compared to the smooth wall: $\Delta B = B_{\text{rough}} - B_{\text{smooth}}$.

It is difficult to construct a grid which results in the desired $y^+$ values for all grid point wall bounded flows. Therefore a scalable wall function would come in handy. In order to have a scalable wall function one should have an expression for $u^+$ and $y^+$ which is valid for all regions. Spalding proposed (Spalding, 1961) or see for example (Villiers, 2006):

$$y^+ = u^+ + \frac{1}{E} \left[ e^{\kappa u^+} - 1 - \kappa u^+ - \frac{1}{2} \left( \kappa u^+ \right)^2 - \frac{1}{6} \left( \kappa u^+ \right)^3 \right]$$

(3.40)

### 3.5.1 Roughness

As read in previous sections roughness does play an important role in the boundary layer theorem. It can be read in (Schlichting, 1979, p.626) that one can distinguish three regions:

- **Hydraulically smooth** regime: the protrusions do not exceed the viscous sub-layer, there for one can expect that the resistance is mainly the result of the laminar shear stress. The following thus will hold:

$$0 \leq \frac{k_s u_T}{\nu} \leq 5$$

(3.41)
• **Transition** regime: protrusions extend partly outside the laminar sub layer and the resistance will not only be from viscous stresses.

\[
5 \leq \frac{k_s u_\tau}{\nu} \leq 70 \tag{3.42}
\]

• **Hydraulically rough** regime: here the protrusions go beyond the sub-layer and will form the largest part of the resistance on the flow.

\[
\frac{k_s u_\tau}{\nu} > 70 \tag{3.43}
\]

Also note that in many literature the ratio \( k_s u_\tau / \nu \) is referred to as the non-dimensional sand roughness:

\[
k_s^+ = \frac{k_s u_\tau}{\nu} \tag{3.44}
\]

In (Schlichting et al., 2000) it is shown that for large roughness heights:

\[
B = 8.0 - \frac{1}{\kappa} \ln \left( \frac{k_s u_\tau}{\nu} \right) \tag{3.45}
\]

In equation above \( k_e \) and \( k_s \) respectively represent the absolute (or effective) wall roughness and equivalent (or Nikuradse) sand roughness, also suggested by (Schlichting, 1979) the following relation:

\[
k_s = 4k_e \approx 2d_{50} \tag{3.46}
\]

The median grain diameter \( d_{50} \) is used to estimate the effective roughness height (half of the sand particle diameter).

Looking at Eq. 3.45 one can see that the parameter \( B \) and thus \( E \) depends on the shear velocity. Another wall function is mentioned in (Blocken et al., 2007) taken from Ansys is. This wall function uses an roughness modification of the smooth wall:

\[
u^+ = \frac{1}{\kappa} \ln \left( \frac{E_y^+}{1 + C_s k_s^+} \right) \tag{3.47}
\]

In Eq. 3.47 \( E \) should be chosen as the hydraulically smooth situation (\( E = 9.8 \)) However the roughness \( C_s \) constant is unclear, it should lay in the interval \([0; 1]\) and fully rough flows \( C_s > 0.2 \). Also if reading the Ansys manual which is referred to in Blocken it is found that the turbulent eddy viscosity should be limited and that if fully rough the viscous layer should be discarded. One can also substitute Eq. 3.45 into Eq. 3.39 and get:

\[
\frac{U}{u_\tau} = \frac{1}{\kappa} \ln \left( \frac{e^{8.0n_w}}{k_s} \right) \tag{3.48}
\]

Similar as in (van Rhee, 2002) one can now write the wall shear stress as:

\[
\tau_{wall} = \frac{\rho_{wall} U_{n_w}^2}{\left( \frac{1}{\kappa} \ln \left( \frac{e^{8.0n_w}}{k_s} \right) \right)^2} \tag{3.49}
\]

The velocity \( U_{n_w} \) denotes the tangent velocity at distance \( n \).
CHAPTER 3. COMPUTATIONAL FLUID DYNAMICS

3.6 Boundary Conditions

3.6.1 Inlet

In finding a numerical solution the boundary conditions of the model domain should be specified, the inlet are discussed in this section. The turbulence intensity $I$ can be estimated for turbulent flows in pipes at $1 - 5\%$. The specific eddy dissipation rate can be estimated using:

$$\omega = \frac{\sqrt{k}}{\ell}$$  \hspace{1cm} (3.50)

The dissipation rate can be estimated using the auxiliary relation form the standard $k - \epsilon$ model:

$$\epsilon = C_\mu \frac{k^{3/2}}{\ell}$$  \hspace{1cm} (3.51)

Presented in (?) it is given that for fully developed pipe flows the length $\ell$ can be estimated using:

$$\ell = 0.038 D_h$$  \hspace{1cm} (3.52)

The term $D_h$ is the hydraulic diameter. This approximation could be used to calculate the initial conditions at the inlet of the model. The model inlet and outlet conditions should be specified. For a high Reynolds flow the turbulence intensity $I$ can be estimated at 5-20\% (?). Using Eq. 3.13 and Eq. 3.14 we can express the turbulent energy at our inlet as:

$$k_{inlet} = \frac{3}{2} (IU_x)^2$$  \hspace{1cm} (3.53)

In above the velocity is assumed to be perpendicular to the surface so only velocity component $(U_x)$ is remaining.

3.6.2 Surface boundary conditions

One also needs to specify the quantity at the surface wall, this can be achieved using the wall function discussed in Eq. 3.5. If the friction velocity is known by applying Eq. 3.38b or 3.39. In (Wilcox, 2010) it is given that the wall functions for $k$, $\omega$ and $\epsilon$ can be expressed as:

$$k = \frac{u_T^2}{\sqrt{\beta}}$$, \hspace{0.5cm} $$\omega = \frac{k^{1/2}}{\beta y^{1/4} k y}$$, \hspace{0.5cm} $$\epsilon = \beta y^{3/4} k^{3/2}$$  \hspace{1cm} (3.54)

One should remember that these equations arise from assuming that the first grid cells are located in the log layer. It is also given by Wilcox that this approach gives numerical solutions which are sensitive to the point where the wall functions are used, mainly separated flows are prone to this. A solution would be replacing the wall functions with ones where the pressure gradient is included, however these are not given here and even not considered in the simulation.
3.6.3 Surface Roughness

For the $k-\omega$ turbulence model (and of course also the $k-\omega\text{SST}$) there surface roughness can be implemented rather easy, the surface boundary condition for $\omega$ is (Hellsten, 1997) or (Wilcox, 1988):

$$\omega_w = \frac{u^*_w}{\nu}S_R$$ (3.55)

The subscript $w$ denotes the position where the distance to the (nearest) wall is $n_w = 0$ and $S_R$ is a non-dimensional function for this rough-wall method and can be calculated using:

$$S_R = \begin{cases} 
\left(\frac{200}{k_s^+}\right)^2 & \text{if } k_s^+ \leq 5 \\
\frac{100}{k_s^+} + \left[\left(\frac{200}{k_s^+}\right)^2 - 100\right]e^{5-k_s^+} & \text{if } k_s^+ > 5 
\end{cases}$$ (3.56)

This boundary condition is unfortunately not implemented in OpenFOAM. Note that there is also a wall boundary condition for porous surfaces, which can be looked up in Wilcox.

3.7 Mixture flows

3.7.1 Viscosity of slurry

The viscosity of a fluid is normally a function of the temperature. For slurries the viscosity is also influenced by the concentration of particles in the fluid. A theoretical model is given by Einstein which is only valid for very low concentrations of solid particles (volumetric concentration of $c_v < 3\%$) and several empirical relations are known. One of them is found in (Thomas, 1965):

$$\mu_r = \frac{\mu_s}{\mu_f} = 1 + 2.5\phi_s + 10.05\phi_s^2 + 0.00273e^{16.6\phi_s}$$ (3.57)

where $\mu_r, \mu_s$ and $\mu_f$ represent respectively the relative, slurry and fluid viscosity. The volume fraction is denoted by $\phi_s$.

3.7.2 Drift flux model

The drift flux model is simply said looking at as the dispersed phases and continuous phase as a mixture, so only one continuous phase is considered. The work of (Goeree et al., 2013) is used as a base to implement this in OpenFOAM and is named as the driftFluxFoam solver, the basic of the model is explained in Appendix B.

3.8 Computational Fluid Dynamics

Because of limiting computer power in the early stage of CFD calculations models where used to simplify or model the turbulence flow charaesistics, it uses the so called Reynolds
Averaged Navier Stokes equations or RANS. When looking at these equations one can see the so called Reynolds stresses which are unknown and models are introduced in order to find a solution. These models are for example k-ε and k-ω which uses the Boussinesq approach: introducing turbulent viscosity. Another way is solving the RANS stresses using so called RSM models but this will introduce more equations to solve. The latter means that more CPU time is needed compared to the other turbulence models.

A more detailed approach is done in the Large eddy simulations or LES. The LES model uses a filter to ‘select’ scales of eddies which are solved. The smaller length scales are not solved but modelled. This model requires a finer grid than RANS and usually requires more computer capacity and calculation time.

If one wants to solve all length scales the method is called DNS, which stands for Direct numerical simulation which uses no (or minimal) models creating a possibility to actually simulate flow behaviours but is experienced to be difficult to use with complex flows.

For DNS the mesh points and thus the number of arithmetic operations will scale with \( n^3 \Delta t^{9/4} \) (\( n^3 \Delta t^3 \) if time is also considered). LES with \( n^3 \Delta t^{3/2} \) (\( n^3 \Delta t^{9/4} \) if also time is consider). Therefore these modelling types are not even considered in this thesis because the Reynolds numbers of interest are in the order of \( 10^5 - 10^6 \) and the so called RANS modelling is used.

Numerical (or false) diffusion can cause significant errors, it occurs for example using the upwind discretisation scheme and can be shown using an case where the (physical) diffusion is set to 0. It occurs for flows with directions not orthogonal to a Cartesian grid. Generally high Reynolds number flows and coarse grids are prone to this numerical difficulty, a solution can be using other discretisation schemes or meshing a finer grid (Versteeg and Malalasekera, 1995).

### 3.8.1 Algorithms

#### 3.8.1.1 SIMPLE algorithm

In this report the SIMPLE\(^4\) algorithm is used to solve the single phase steady state incompressible flow.

- The algorithm starts by an initial guess of the pressure field \( p^* \),
- The velocity field \( v^* \) in all directions are then solved by using the momentum equations,
- Then a correction step is applied by calculating the error which arise from guessing the pressure and velocities, this is done using the so called pressure correction equation
- With the correction the pressure and velocities can be corrected
- Now all other quantities are solved
- Until converged, repeat these steps using for each loop the newly calculated values as the next guess.

\(^4\)Semi-Implicit Method for Pressure-Linked Equations, see (Versteeg and Malalasekera, 1995)
3.8.1.2 PISO algorithm

In this report the PISO\(^5\) algorithm is used to solve the single phase, unsteady state and incompressible flow. It can also be used for compressible flows! it is basically an extension of the SIMPLE algorithm: one more correction step is done. This means an extra set of pressure correction equation is solved to ensure that the continuity equation holds.

3.8.2 OpenFOAM

The simulations for this thesis are done using OpenFOAM\(^6\). In this section some relevant aspects of this software package are discussed now.

3.8.2.1 Solvers

Many solvers are available in the used OpenFOAM distribution, the ones used are shortly mentioned here. The SIMPLE algorithm can be used by choosing the simpleFoam solver where the PISO algorithm implementation can be used by choosing the pisoFoam solver. Both algorithms were briefly explained earlier, one more noteworthy solver is the driftFluxFoam solver which is briefly explained now.

**driftFluxFoam solver / algorithm**

The driftFluxFoam solver uses the mixture approach for solving 2 incompressible fluids, it uses the drift-flux approximation for determining the relative motion between the two phases. It can be used for simulating the settling of the dispersed phase. The solver uses the PIMPLE algorithm with an extra pressure-velocity correction step, it is from the PISO and SIMPLE algorithms. However further investigation of the implementation of this solver in the OpenFOAM is not done during this project.

Rather a limitation of the implementation is that only k-\(\varepsilon\) (and its buoyant version) turbulence model is available, this makes it (not yet) possible to use for example k-\(\omega\) or k-\(\omega\)SST turbulence model.

3.8.2.2 Boundary conditions

In OpenFOAM one can specify boundary conditions to the so called patch fields. If one wants to specify values at te boundary which should be kept constant over all calculation steps the fixedValue should be used. It can be used to specify scalars or vectors over the complete patch.

Another common boundary conditions is specifying the gradient on the patch, and if this gradient is zero then one can use the zeroGradient condition. Otherwise the fixedGradient and the desired gradient should be applied and specified.

\(^5\)Pressure Implicit with Splitting of Operators, see (Versteeg and Malalasekera, 1995)

\(^6\)The OpenFOAM version used in this thesis is 2.4.0.
The eddy viscosity is defined with the relevant eddy viscosity equation, depending on which turbulence model is used (for example for \( k-\varepsilon \) this is Eq. A.2). So the eddy viscosity can be calculated with \( k \) and \( \epsilon \) (or \( k \) and \( \omega \)). Because of this relation sometimes the eddy viscosity should just be ‘calculated’, to do so the calculated is used.

More ‘complex’ boundary conditions and wall functions available in the OpenFOAM distribution are discussed briefly now, mostly the implementation is based on earlier discussed wall functions.

**kq\( \overline{R} \)WallFunction**

This boundary condition provides a suitable specification for turbulence \( k \), \( q \), and \( R \) fields for the case of high Reynolds number flow using wall functions. It is a simple wrapper around the zero-gradient condition.

**kLowReWallFunction**

This boundary condition provides a turbulence kinetic energy wall function condition for low- and high-Reynolds number turbulent flow cases. The model operates in two modes, based on the computed laminar-to-turbulent switch-over \( y^+_\text{lam} \) value derived from kappa and \( E \) same as Eq. 3.62. It also calculates the shear velocity by:

\[
\frac{u_\tau}{\nu} = C_0^{0.25} \sqrt{k} \quad (3.58)
\]

Then the cells \( y^+ \) is calculated by:

\[
y^+ = \frac{u_\tau n_w}{\nu} \quad (3.59)
\]

Finally \( k \) is calculated using:

\[
\frac{k}{u_\tau^2} = \frac{2400.0}{(C_\varepsilon 2)^2} \left( \frac{1}{(y^+ + 11.0)^2} + \frac{2y^+}{11.0^3} - \frac{1}{11.0^2} \right) \quad \text{for: } y^+ \leq y^+_\text{lam} \quad (3.60a)
\]

\[
\frac{k}{u_\tau^2} = \frac{-0.416}{\kappa} \log(y^+) + 8.366 \quad \text{for: } y^+ > y^+_\text{lam} \quad (3.60b)
\]

**epsilonWallFunction**

This boundary condition provides a turbulence dissipation wall function condition for high Reynolds number turbulent flow cases. The condition can be applied to wall boundaries, whereby it:

- calculates \( \epsilon \) and \( G \) (turbulence generation field)
- inserts near wall \( \epsilon \) values directly into the \( \epsilon \) equation to act as a constraint

The values are calculated by:

\[
\epsilon = \frac{C_\mu^{0.75} k^{1.5}}{\kappa n_w} \quad (3.61)
\]
which in fact is the expression for $\epsilon$ found in Eq. 3.54 where one should remember that $\beta^* = C_\mu = 9/100 = 0.09$.

**epsilonLowReWallFunction**

This boundary condition provides a turbulence dissipation wall function for low- and high-Reynolds number turbulent flow cases. The condition can be applied to wall boundaries, whereby it inserts near wall epsilon values directly into the $\epsilon$ equation to act as a constraint. The model operates in two modes, based on the computed laminar-to-turbulent switch-over $y_{tlam}$ value derived from $\kappa$ and $E$:

$$y_{tlam}^+ = \log(\max(Ey^+, 1))/\kappa \quad (3.62)$$

The cell $y^+$ value for is calculated by using:

$$y^+ = C_{1/4}^{1/4} \frac{\sqrt{\kappa n_w}}{\nu}; \quad (3.63)$$

The value for epsilon is then calculated by:

$$\epsilon = \begin{cases} 
\frac{2k\nu}{n_w^2} & \text{for: } y^+ \leq y_{tlam}^+ \quad (3.64a) \\
C_\mu^{0.75}k^{1.5} & \text{for: } y^+ > y_{tlam}^+ \quad (3.64b)
\end{cases}$$

**omegaWallFunction**

This boundary condition provides a wall function constraint on turbulence specific dissipation, $\omega$. The values are computed using:

$$\omega = \sqrt{\omega_{vis}^2 + \omega_{log}^2} \quad (3.65)$$

where $\omega_{vis}$ is calculated using:

$$\omega_{vis} = \frac{6.0\nu}{\beta_1 n_w^2} \quad (3.66)$$

and where $\omega_{log}$ is calculated using:

$$\omega_{log} = \frac{\sqrt{k}}{(C_\mu)^{0.25} \kappa n_w}; \quad (3.67)$$

The implementation of this in OpenFOAM lets you specify the parameter $E$, which is used together with the $\nu_t$ boundary condition to determine the value $y_{PlusLam} (y_{tlam}^+)$ although further use of the calculated value is not found in the source code. It is believed to be needed for calculating the turbulent viscosity so the parameter should correspond with all other set boundary conditions.
CHAPTER 3. COMPUTATIONAL FLUID DYNAMICS

**nutUSpaldingWallFunction**

This boundary condition provides a turbulent kinematic viscosity condition when using wall functions for rough walls, based on velocity, using Spalding’s law to give a continuous nut profile to the wall ($y^+ = 0$)

$$y^+ = u^+ + \frac{1}{E} \left[ e^{\kappa u^+} - 1 - \kappa u^+ - \frac{1}{2} (\kappa u^+)^2 - \frac{1}{6} (\kappa u^+)^3 \right]$$  \hspace{1cm} (3.68)

The turbulent eddy viscosity is calculated using the Newton-Rhapson method and the velocity of the first grid point.

**nutkRoughWallFunction**

Looking at the source code it is unclear what actually happens, and what value for $C_s$ should be chosen. It is believed that the code implements Eq. 3.47.

This boundary condition provides a turbulent kinematic viscosity condition when using wall functions for rough walls, based on turbulence kinetic energy. The condition manipulates the $E$ parameter to account for roughness effects. Required inputs are sand-grain roughness height $k_s$ and roughness constant $C_s$.

It calculates

$$u^+ = C_0^{0.25} \sqrt{k}$$  \hspace{1cm} (3.69a)

$$y^+ = \frac{u^+ n_w}{\nu}$$  \hspace{1cm} (3.69b)

$$k_s^+ = \frac{u^+ k_s}{\nu}$$  \hspace{1cm} (3.69c)

Where the viscosity of the fluid is taken. It modifies $E$ with

$$E' = \frac{E}{S_E}$$  \hspace{1cm} (3.70)

with

$$S_E = \left( \frac{k_s^+ - 2.25}{87.75} + C_s k_s^+ \right) \sin(0.4258 \log(k_s^+) - 0.811)$$ \hspace{1cm} for: $k_s^+ < 90$  \hspace{1cm} (3.71a)

$$S_E = 1.0 + C_s k_s^+$$ \hspace{1cm} for: $k_s^+ \geq 90$  \hspace{1cm} (3.71b)

$$\nu_t = \max \left( \min \left( \frac{\nu y^+ k}{\log(\max(E'y^+, 1 + 1 \cdot 10^{-4})) - 1}, 2\nu_t, 0.5\nu_t \right), 0.5\nu_t \right)$$  \hspace{1cm} (3.72)

where

$$\nu_t, \text{lim} = \max (\nu_t, \nu)$$  \hspace{1cm} (3.73)
nutURoughWallFunction

This boundary condition provides a turbulent kinematic viscosity condition when using wall functions for rough walls, based on velocity. It uses the roughness height \( k_s \), the roughness constant \( C_s \) and a roughness (or scaling) factor \( f \). Complete implementation in OpenFOAM is not given here because this function is not used in this project. It is recommended when one wants to use this boundary condition the literature should be sought were the implementation is based on.

3.8.2.3 Inlet

Several possibilities are available for inlet specification in OpenFOAM, the ones used in this thesis are discussed now.

\textit{turbulentMixingLengthDissipationRateInlet}

This boundary condition provides a turbulence dissipation \( \epsilon \) inlet condition based on a specified mixing length. The patch values are calculated using:
\[
\epsilon = \frac{C_0 \mu k^{1.5}}{\ell} \tag{3.74}
\]

This actually differs from Eq. 3.51, however it is still an estimation of the inlet value so no major impact is expected.

\textit{turbulentIntensityKineticEnergyInlet}

This boundary condition provides a turbulent kinetic energy condition, based on user-supplied turbulence intensity, defined as a fraction of the mean velocity. It calculates the turbulent kinetic energy \( k \) with:
\[
k = \frac{3}{2} I U_{\text{mag}}^2 \tag{3.75}
\]

Which actually is Eq. 3.53 discussed in earlier in this report but without the velocity simplification.
Chapter 4

Validation OpenFOAM

4.1 Introduction

As discussed in the previous chapter the OpenFOAM code is validated now. This is done using three cases based on published reports. First the experiments from (Eriksson et al., 1998) is used. Then roughness effects are validated using the experiments from (Rostamy et al., 2010) and last the higher Reynolds number case is simulated using the (Förthmann, 1934) experiments.

The plane wall jet is used widely as validation cases and the numerical solutions are published thoroughly. For example, Eriksson and Abrahamson published an article in the knowledge base of ERCOFTAC (Eriksson and Karlsson, 2015) briefly explaining their experiments to create a validation case for simulating the plane wall jet. A workshop resulted in 19 solutions for the plane wall jet where different turbulence models are used on a variety of grid sizes. The mainly used turbulence models were k-ε and RSM models.

Although not giving a clear explanation why, the k-ε models where not giving good result when wall functions are applied. The skin friction / wall shear stress results are 20-30 % higher than the experimental determined values. The best simulation results were done by using the RSM models. However the Drift Flux model is not implemented in OpenFOAM together with RSM models. Also the RSM models introduce more equations which increase calculation time and the RSM model known to have stability issues. So the RSM model is not investigated.

Also given in the ERCOFTAC article is the suggestion of using an ‘physical more appealing near-wall model formulation’. It is this reason why Menters’ k-ωSST model is used, which is known to produce good results for (semi) confined flows, although this turbulence model is not compatible with the Drift Flux model in OpenFOAM .

The boundary conditions suggested are given now: For the velocity should be taken the zero slip condition on the solid surfaces, and an developed profile at inlet. All other, like turbulent quantities are suggested to be taken as the experiments. This however is not done in the simulation, solely because difficulties arise from the limitations of the knowledge / know how of OpenFOAM. Wall Functions are believed to be a reason of the faulty results, therefore resolved grid are needed with the necessary boundary conditions.
Also in order to test some OpenFOAM code an open channel simulation is done, the results are shortly discussed in Appendix E. Now the general set-up of the cases simulated are discussed, after that the specific details of each case are evaluated. The cases simulated are the experiments from Eriksson (see Section 4.3.1), Rostamy (see Section 4.3.2) and Förthmann (see Section 4.3.3). The first two experiments were conducted using water as medium, the last was conducted using air.

4.2 General

All plane wall jet simulations follow a general set-up discussed now and different grid sizes are used. A close up of a typical grid layout is given in Figure 4.2. The grids are build using the OpenFOAM blockMesh, which is briefly explained in Appendix D, all grids used are given in Table 4.1. The near wall region has a finer grid in order to solve the near wall flow properties, it has the same height as the slot $b_0$. The region above this has generally more cells but is much larger in size as seen in Figure 4.2. The boundary conditions are given in now, the face names are shown in Figure 4.1
Figure 4.2: A close up of a typical grid layout, also shown is the near wall region where a higher refinement is chosen as the far wall region.

Table 4.1: Grid specifications used for simulating the plane wall jet, for the near wall region and far from region the number of cells in x,y,z direction are given, with the expansion factor $R$ for the grid grading towards the bottom marked with blue as explained in Appendix D.

<table>
<thead>
<tr>
<th>Name</th>
<th>L</th>
<th>H</th>
<th>Near wall region</th>
<th>Far from wall region</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid A</td>
<td>2</td>
<td>1.5</td>
<td>500 1 30</td>
<td>500 1 470</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>1</td>
<td>2 2 2</td>
<td>1 2</td>
</tr>
<tr>
<td>Grid B</td>
<td>2</td>
<td>1.5</td>
<td>200 1 10</td>
<td>200 1 200</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>1</td>
<td>2 2 2</td>
<td>1 8</td>
</tr>
<tr>
<td>Grid C</td>
<td>2</td>
<td>1.5</td>
<td>150 1 2</td>
<td>150 1 100</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>1</td>
<td>2 2 2</td>
<td>1 8</td>
</tr>
<tr>
<td>Grid D</td>
<td>4</td>
<td>2</td>
<td>400 1 20</td>
<td>400 1 200</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>1</td>
<td>2 2 2</td>
<td>1 6</td>
</tr>
<tr>
<td>Grid E</td>
<td>4</td>
<td>2</td>
<td>200 1 4</td>
<td>200 1 150</td>
</tr>
<tr>
<td>(not used)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>1</td>
<td>1 10</td>
<td>1 15</td>
</tr>
<tr>
<td>Grid F</td>
<td>4</td>
<td>2</td>
<td>400 1 20</td>
<td>100 1 200</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>1</td>
<td>2 10</td>
<td>1 15</td>
</tr>
</tbody>
</table>

45
CHAPTER 4. VALIDATION OPENFOAM

4.2.1 Boundary conditions

Table 4.2: The boundary conditions used for all plane wall jet models using both the $k - \epsilon$ or $k - \omega_{SST}$ turbulence model

<table>
<thead>
<tr>
<th>Patch</th>
<th>Field</th>
<th>Type</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>inlet</td>
<td>$U$</td>
<td>fixedValue</td>
<td>inlet profile</td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>turbulentIntensityKineticEnergyInlet</td>
<td>intensity 0.01</td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>calculated</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\epsilon$</td>
<td>turbulentMixingLengthDissipationRateInlet</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>fixedValue</td>
<td>uniform $1 \cdot 10^{-15}$</td>
</tr>
<tr>
<td>outlet</td>
<td>$U$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>fixedValue</td>
<td>uniform 0</td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>calculated</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\epsilon$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td>bottom</td>
<td>$U$</td>
<td>fixedValue</td>
<td>uniform $(0 \ 0 \ 0)$</td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>nutUSpaldingWallFunction</td>
<td></td>
</tr>
<tr>
<td>top</td>
<td>$U$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>fixedValue</td>
<td>uniform 0</td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>calculated</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\epsilon$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td>other</td>
<td>$U$</td>
<td>fixedValue</td>
<td>uniform $(0 \ 0 \ 0)$</td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>nutUSpaldingWallFunction</td>
<td></td>
</tr>
<tr>
<td>front</td>
<td>all</td>
<td>empty</td>
<td></td>
</tr>
<tr>
<td>back</td>
<td>all</td>
<td>empty</td>
<td></td>
</tr>
</tbody>
</table>
The low Reynolds wall functions in OpenFOAM chosen are given in Table 4.3. Furthermore this table shows the high Reynolds wall functions that are also used.

Table 4.3: The boundary conditions applying for the plane wall jet models using the $k - \epsilon$ or $k - \omega_{SST}$ turbulence model and low and high Reynolds wall functions

<table>
<thead>
<tr>
<th>Patch</th>
<th>Field</th>
<th>Type</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>Low Reynolds Wall Functions:</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>bottom</td>
<td>$k$</td>
<td>kLowReWallFunction</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\epsilon$</td>
<td>epsilonLowReWallFunction</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>omegaWallFunction</td>
<td></td>
</tr>
<tr>
<td>other</td>
<td>$k$</td>
<td>kLowReWallFunction</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\epsilon$</td>
<td>epsilonLowReWallFunction</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>omegaWallFunction</td>
<td></td>
</tr>
<tr>
<td>High Reynolds Wall Functions:</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>bottom</td>
<td>$k$</td>
<td>kqRWallFunction</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\epsilon$</td>
<td>epsilonWallFunction</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>omegaWallFunction</td>
<td></td>
</tr>
<tr>
<td>other</td>
<td>$k$</td>
<td>kqRWallFunction</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\epsilon$</td>
<td>epsilonWallFunction</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>omegaWallFunction</td>
<td></td>
</tr>
</tbody>
</table>
4.3 Validation Cases

As discussed earlier, many numerical solutions are presented based on the standard k-\(\varepsilon\) turbulence model, however, no work is found by the writer of this thesis that uses the k-\(\omega\)SST model. Because of this reason and the fact that the recommendations found earlier in this chapter that this model is tested. To create a comparison, the standard k-\(\varepsilon\) model is also used in several cases. In order to test the wall functions that are implemented in OpenFOAM, it is chosen that for each turbulence model 2 (low and high) different sets of solutions are created based on the validity in which layer the first grid point is located.

4.3.1 Eriksson

Eriksson conducted and published experiments of the two-dimensional plane turbulent wall jet in (Eriksson et al., 1998), the experiments were done with \(Re_0 \approx 9.6 \times 10^3\) performed in water as medium and the inlet velocity \(U_0 \approx 1.0\, m/s\) with a slot height \(b_0 \approx 9.6\, mm\). Inlet conditions where optimised (?) using a large contraction with a turbulence-reducing screen so flat mean velocity profile would appear at the inlet, the velocity was kept constant by introducing two wears to make use of the relation \(U_0 = \sqrt{2g\Delta h}\). The measurements were done using LDV\(^1\). Measurements were done at the non-dimensional distances \(x/b_0 = \{5, 10, 20, 40, 70, 100, 200\}\). Some of the turbulence correlated quantities were measured and also the turbulence intensity was measured to be less than 1%. In (Eriksson et al., 1998) the evolution of the turbulence quantities are given for \(x/b_0 = \{5, 10, 20, 40\}\), where the transition of laminar to turbulent flow is seen here. The linearised dimensionless growth rate of the jet does not coincide with the introduced values in (Launder and Rodi, 1981) and (Rajaratnam, 1976) but it was found that the jet growth follows:

\[
\frac{b}{b_0} = 0.0782\frac{x}{b_0} + 0.332
\]  

(4.1)

The empirical fitted relation for the skin friction found:

\[
c_f = 0.0179/Re_0^{0.113}
\]  

(4.2)

The velocity scale decay measurements in the report were plotted as \(\log_{10}(U/m)\) versus \(\log_{10}(b)\), so together with the growth rate of the jet it could be replotted as shown in Figures 4.3a and 4.3b. The constant \(C_1\) is determined by using Eq. 2.7 each of the measured points and take the average value of those.

4.3.1.1 Simulation

Because inlet conditions where measured and published by Eriksson the CFD model is build without an inlet tube to create a developed inlet flow. Instead the velocity profile

\(^1\)Laser Doppler Velocimetry, can be used to measure velocities in (semi-) transparent fluid flows
is specified at the inlet. Referring to Figure 4.1 the length is chosen to be $L = 2m$ and the height $H = 1.5m$, the width is set to $W = 0.1m$ but in this 2D model quantities in y-direction are not solved. Using the RANS model it is not possible to determine the individual fluctuations so it is not possible to check with the measured quantities. Only the Eriksson experiments gave some insight in these fluctuations. Using these, it should be possible to determine the kinetic turbulent energy of the flow. However the turbulent kinematic energies are not considered in this thesis.

4.3.1.2 Results

The Eriksson based CFD model results are discussed now. This experiment is evaluated most extensive in this thesis. The two turbulence models are used ($k$-ε and $k$-$\omega$SST ) for three grid refinements and their relevant boundary conditions. The best results were obtained for the middle refined grid.

As expected the $k$-$\epsilon$ model results in an over-prediction of the shear stress. This can clearly seen not only from the shear stress plot but also from the maximum velocity decay plots in Figure 4.3a, where the flow tends towards the rough surface plane wall jets despite the fact that a smooth wall is given as a boundary condition. The $k$-$\omega$SST models produce better results for all grid sizes, except the coarsest grid will result in 'strange' wiggles near the inlet shown in 4.3b.

The velocity scale growth from the simulation are best for the $k$-$\omega$SST as seen in Figure 4.3b, the $k$-$\epsilon$ model show an over prediction of the growth as seen in Figure 4.4a.

The friction coefficient versus the local Reynolds number are shown for the $k$-$\epsilon$ model in Figure 4.5a and for the $k$-$\omega$SST in Figure. Again $k$-$\omega$SST 4.5b produce the best results, and even the Sigalla friction coefficient relation shown in Eq. 2.14 correspond.

The $y^+$ values of the first grid cells are calculated in MATLAB using an iterative solving method and applying the Spalding Single Formula. Because of exporting is done using ParaView the values calculated are the first points above the wall, the $y^+$ values calculated from OpenFOAM differ from it, mainly because it is not clear how OpenFOAM calculates the velocity gradient at the wall. The calculated $y^+$ values of the first grid points can be found in Appendix ??.

Good to note is that the low and high Reynolds number wall functions produce almost similar results on the middle grid. The offset for the middle grid solution is believed to be caused by the lack of developed inlet conditions. The dimensionless velocity profiles (turbulent boundary layer) are given in Figures 4.6a and 4.6b for only the Grid B$^2$, here one can see that again the $k$-$\omega$SST on the middle grid produce the best results. The peak (blue) in the $k$-$\epsilon$ is present probably because of the sudden growth seen in Figure 4.4a.

---

$^2$The other simulations results are omitted here, this is done for the sake of clarity. All results can be looked up in the Appendix ??.
Figure 4.3: Comparison of maximum velocity decay for the Eriksson case: OpenFOAM simulation results are shown for different grid sizes and wall functions. (a) are calculated using the $k$-$\epsilon$ and (b) the $k$-$\omega$SST turbulence model. Also shown is the maximum velocity decay equation Eq. 2.7 fitted on the experimental data from (Eriksson et al., 1998) by $C_1 = 3.55$. 

50
Figure 4.4: Comparison of the scale growth ($b$) for the Eriksson case: OpenFOAM simulation results are shown for different grid sizes and wall functions, (a) are calculated using the $k$-$\epsilon$ and (b) the $k$-$\omega$SST turbulence model. Also shown is the experimental data from (Eriksson et al., 1998).
Figure 4.5: Friction coefficient $c_f$ versus the local Reynolds Number $Re_m$ for (a) k-$\epsilon$ and (b) k-$\omega$SST model. Also shown are experimental results from Eriksson and Abrahamsson taken from (Eriksson et al., 1998), together with Sigalla’s relation shown in Eq. 2.14.
Figure 4.6: The near wall velocity profiles calculated from the simulation results for the Eriksson case with (a) k-\(\epsilon\) and (b) k-\(\omega\)SST turbulence model on grid B and the high Reynolds wall functions shown as the continuous lines. Together with the Eriksson experiment and the Spaldings law of the wall function.
4.3.2 Rostamy

In the report of Rostamy (Rostamy et al., 2010) the effects of roughness on the wall jet flow characteristics were investigated, using LDV. The nozzle Reynolds number was estimated to be 7500 and a non-dimensional roughness of $5 \leq k_s^+ \leq 70$ was created using a 36-grit sheet. The conclusion is that surface roughness does not affect the spread rate or the decay of the maximum velocity, however the thickness of the inner layer increases. The roughness caused an increase of about 30% (at $Re_m \approx 3500$). The inlet velocity used for the experiment was $U_0 \approx 1.21m/s$ and the slot height $b_0 = 6mm$. Same as Eriksson the turbulence intensity at the inlet was less than 1%. The velocities where measured up to $x/b_0 = 80$. The resulting skin friction coefficients versus the local Reynolds number can be found in figure 2.9.

In the published report the inner scale velocity profiles and the near wall velocity distribution (dimensionless) are given. The non-dimensional sand roughness $k_s$ was estimated by the writers to be 25. Which is justified when using Eq. 3.45 and plotting Eq. 3.38b with the value obtained ($B = 0.1491$ and thus $E = 1.063$). These are shown in Figure 4.7.

![Figure 4.7](image-url)

Figure 4.7: The inner velocity scale of the smooth (Eriksson) and rough (Rostamy) turbulent wall jet, nondimensionalised for different nozzle Reynolds numbers. Data digitalised from (Rostamy et al., 2010) and (Eriksson et al., 1998). The log wall function Eq. 3.38b and Spaldings’ wall function Eq. 3.40 are also plotted for the calculated $B$ and $E$ using Eq. 3.45 for $k_s^+ \approx 25$ and $\kappa$ set to the standard value $\kappa = 0.41$. 

54
4.3.2.1 Simulation

Because the simulation parameters, except for the roughness, do not differ much from the Eriksson case the domain has the same sizes as that case ($L = 2m$, $W = 0.1m$ and $H = 1.5m$). The dimensionless roughness height $k_s^+$ corresponds to $E = 1.063$. Only the high-Reynolds number wall functions are used and on only one grid layout is used (Grid C) which resulted in the correct wall distances so that the first grid points lay in the logarithmic layer.

4.3.2.2 Results

The $k$-epsilon model again over predicted the shear stress the $k$-$\omega$SST model showed good results. The scaled velocity profiles had the best shape for the $k$-$\omega$SST model. The maximum velocity scale decay followed the curve very well for $C_1 = 3.2$ which is shown in Figure 4.10, despite that this figure is not so useful because of no experimental data is present, it is good to show the shape of the curves. The velocity scale growth is omitted here but is shown Appendix ??.

Also the dimensionless velocity $u^+$ versus the non dimensional wall distance curves $y^+$ are plotted in Figures 4.8a and 4.8b: the results comply best with the experimental data for the $k$-$\omega$SST model. The friction factors in Figure 4.9 do not coincide with the presented experimental data however one can easily see that it lays on the same “line” that runs parallel to Sigalla’s relation (Eq. 2.14), this however does not hold for the $k$-$\epsilon$ model.
Figure 4.8: The near wall velocity profiles calculated from the simulation results for the Rostamy case with (a) k-\( \epsilon \) and (b) k-\( \omega \) SST turbulence model on grid B and the high Reynolds wall functions shown as the continuous lines. Together with the Rostamy experiment and the Spaldings law of the wall function (Eq. 3.40) for \( E = 1.063 \) and \( \kappa = 0.41 \).
Figure 4.9: Friction coefficient $c_f$ versus the local Reynolds Number $Re_{m}$ for $k$-$\epsilon$ and $k$-$\omega$ SST model. Also shown are experimental results from Rostamy and Tachie taken from (Rostamy et al., 2010), together with Sigalla’s relation shown in Eq. 2.14.
Figure 4.10: Comparison of maximum velocity decay for the Rostamy case: OpenFOAM simulation results are shown for different grid sizes and wall functions, calculated using the k-ε and the k-ωSST turbulence model. Also shown is the maximum velocity decay curve for $C_1 = 3.2$ this coefficient is fitted on the simulation data for the k-ωSST model.
4.3.3 Förthmann

According to (Förthmann, 1934) the inlet velocity was assumed to be uniform and equal to 26.5 m/s the medium was air so the kinematic viscosity is guessed to be $\nu \approx 15.11 \cdot 10^{-6}$. The slot height $b_0 = 30 mm$. A plot of the velocity decay is not given in its original report, however from its velocity distributions one can produce it like shown in Figure 4.11. The constant $C_1$ for Eq. 2.7 is determined by averaging each points values (only for points $x/b_0 > 15$). Unfortunately the points do not follow the similarity theory Eq. 2.7. Altering the power term in Eq. 2.7 from 0.5 to 0.4 by applying curve fitting results in:

$$\frac{U_m}{U_0} = \frac{C_1}{(x/b_0)^{0.4}}$$

(4.3)

will give some good resemblance with the experiments. However the validity of these experimental results and of course the Eq. 4.3 are questionable as the CFD results given later on show a better resemblance with Eq. 2.7. The constant $C_1$ is roughly 15% higher than the used value in Rajaratnam and the found value for the Erikssons experiment and it is very unclear if this value is actually correct.

![Figure 4.11: Velocity scales for plane wall jets calculated from (Förthmann, 1934), together with the maximum velocity decay Equation 2.7 for $C_1 = 4.55$ and the modified Equation 4.3 for $C_1 = 3.20$.](image)

4.3.3.1 Simulation

The slot used in the experiments of Förthmann is larger, with respect to the previous 2 experiments discussed, and also a higher nozzle velocity is used. For these reasons the domain is also chosen larger ($L = 4m$, $W = 0.1m$ and $H = 2m$). The surface is believed to be hydraulically smooth therefore the parameter $E = 9.8$ is chosen. Only the high-Reynolds number wall functions are used and only one grid layout is used which resulted in the correct wall distances, so that the first grid points lay in the logarithmic layer.
4.3.3.2 Results

Results are difficult to discuss for this case, the only aspect which can be validated is the decay of the maximum velocity, the velocity scale growth is not given in the original report. Comparing the decay of the maximum velocity is done by looking at Figure 4.13. Again the k-ωSST is the closest to the experimental data, and it is shown that the decay follows the relation in Eq. 2.7 which questions the validity of the experimental data / determined data. Despite there is no data known on the wall shear stress the relation of the local reynolds number and the friction coefficient is given in Figure 4.12 where also the relation introduced by Sigalla is shown. It looks that both turbulence models comply with this. The velocity versus the wall distance (both non-dimensional) is not given here because of no validation data is present, but it can be looked up in Appendix ??.

Note that the k-ε models shows an early wall separation of the flow, therefore the curves stop at a shorter distance from the inlet.

![Figure 4.12: Friction coefficient $c_f$ versus the local Reynolds Number $Re_m$ for k-ε and k-ωSST model for the Förthmann case together with Sigalla’s relation shown in Eq. 2.14.](image)

4.4 Conclusion

In all simulations the k-ωSST turbulence model produced the best results, it should however be noted that at higher flow velocities the differences between the standard k-ε model become smaller. The wall function which apply only for the log region. In
Figure 4.13: Results of the simulation of the Förthmann case 4.13 shows the maximum velocity decay, it shows that the k-ωSST model gives the best results. Note that the k-ε models shows an early wall separation of the flow, therefore the curve stop at a shorter distance from the inlet.

other words, in the correspondence with the used terminology in this report, the high Reynolds number wall functions produce the best results.
Chapter 5

Simulating the flow in the scour hole

5.1 Introduction

In this chapter the model is discussed for the flow in the scour holes. First some cases based on BAGT are simulated for different roughness parameters $E$ in order to get an insight what value is to be chosen. The choice is made after comparison of the obtained velocity profiles with the ones determined experimentally.

After the choice is made which roughness parameter $E$ is best, the modelling of some SSB cases is done. At last the results are evaluated and the erosion velocities obtained from the calculated shear stresses are compared with the kinematic erosion velocities (determined from the shape of the scour hole and its propagation).

The jet model is build up using the OpenFOAM tool snappyHexMesh. It uses STL files to snap on a structured mesh and, if specified creates refinement, towards surfaces or regions. The faces are named as shown in the sketch in Figure 5.1.

5.1.1 Model Decisions

While setting up the simulations it became clear that decisions had to be made, some are compromises. The choices made are summed here:

Deleted

5.1.2 Boundary Conditions

The Boundary conditions are given in Table 5.1, these are used for both the BAGT and SSB cases. The boundary conditions are explained in Section 3.8.2.2.
Figure 5.1: Face names of the jet model, also the dimensions are shown for length $L$, width $W$ and height $H$. 
Table 5.1: The boundary conditions applying for all jet models using the $k - \omega_{SST}$ turbulence model

<table>
<thead>
<tr>
<th>Patch</th>
<th>Field</th>
<th>Type</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>inlet</strong></td>
<td>$U$</td>
<td>fixedValue</td>
<td>uniform ($U_0$)</td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>turbulentIntensityKineticEnergyInlet</td>
<td>intensity 0.01</td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>calculated</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>fixedValue</td>
<td>uniform $1 \cdot 10^{-15}$</td>
</tr>
<tr>
<td><strong>outlet</strong></td>
<td>$U$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>fixedValue</td>
<td>uniform 0</td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td><strong>jet boundary, sand bed</strong></td>
<td>$U$</td>
<td>fixedValue</td>
<td>uniform (0 0 0)</td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>kqRWallFunction</td>
<td>uniform $1 \cdot 10^{-5}$</td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>nutUSpaldingWallFunction</td>
<td>uniform $1 \cdot 10^{-5}$</td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>omegaWallFunction</td>
<td></td>
</tr>
<tr>
<td><strong>top</strong></td>
<td>$U$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>fixedValue</td>
<td>uniform 0</td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td><strong>nozzletube</strong></td>
<td>$U$</td>
<td>fixedValue</td>
<td>uniform (0 0 0)</td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>kqRWallFunction</td>
<td>uniform $1 \cdot 10^{-5}$</td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>nutUSpaldingWallFunction</td>
<td>uniform $1 \cdot 10^{-5}$</td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>omegaWallFunction</td>
<td></td>
</tr>
<tr>
<td><strong>other</strong></td>
<td>$U$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>zeroGradient</td>
<td></td>
</tr>
</tbody>
</table>
CHAPTER 5. SIMULATING THE FLOW IN THE SCOUR HOLE

5.2 BAGT

Because the only velocity measurements documented is available for only one case, and despite its incompleteness, the BAGT experiment discussed in Section 2.1.2 is used to determine the roughness of all other simulations. The roughness parameter is used to try and capture the surface sand injection and possible roughness effects, by choosing the right E-value (roughness) and is determined by calculating three cases discussed shortly: Hydraulically smooth boundary, transitional and a fully hydraulically rough situation.

5.2.1 Hydraulically Smooth Boundary

At first the simulation is run using a hydraulically smooth boundary assumption e.g. $E = 9.8$ for the wall functions. The results are compared with the measured velocity profiles and are shown in Figure ??.

This case is simulated using multiple stages, where each consecutive stage has a finer grid compared to the previous. So starting with finding the solution using the coarsest grid the results are mapped on the next if a converged solution is reached. The mapping is done using the mapFields utility.

This approach enables somewhat faster traject in finding the solution at the correct $y^+$ values at the boundaries. At the first grid cells near the wall the $y^+$ should be between approximately 20 and 200 for solution to be valid. However even after tremendous large domain this was not reached at each grid cell as shown in Figure 5.2a for the bottom values: some peaks exceed the $y^+ = 200$ limit. Further refinement is however not done as the impact of this is assumed to be negligible, and calculation costs would exceed the available resources.

In order to visualize the velocity distributions along the length of the jet body (i.e. in $x$-direction) 8 so called slices are extracted from the simulation results and the velocities are given in Figure ???. Note that only for this case these slices are extracted from the results of the simulations and shown.

5.2.2 Transitional Rough Boundary

The second case, using a transitional rough boundary assumption, is simulated. The roughness is implemented for $k_s^+ = 25$ which results in $E = 1.063$ for the wall functions. Again the results are compared with the measured velocity profiles and are shown in Figure ?? with the same shift in depth as discussed for the smooth boundary. Also here multiple simulations were done each time on a finer grid. And like the smooth boundary simulation peaks exist where the $y^+$ values exceed 200 (Figure 5.2b) but no further refinement is done.

5.2.3 Hydraulically Rough Boundary

Finally the simulation is run using a hydraulically rough boundary: $k_s^+ = 100$ which results in roughness parameter $E = 0.266$. Again The results are compared with the
Figure 5.2: Results of simulations with roughness parameters (see the turbulent boundary layer discussion in Section 3.5) $E = 9.8$, $E = 1.063$ and $E = 0.266$. 
measured velocity profiles and are shown in Figure ?? with the same shift in depth as discussed earlier. Also here multiple simulations were run each time on a finer grid. The $y^+$ values are shown in Figure 5.2c concluding that the $y^+$ values exceed 200 (Figure 5.2b) but no further refinement is done.
5.2.4 Kinematic boundary

Another simulation was done using kinematic boundary conditions, meaning the velocity at the jet boundary and sand bed was defined. The velocity is equal to the trailing speed of the nozzle, only in opposite direction. However the solution was not reached, because of non-convergence, and this path was abandoned after few attempts.

The values for the turbulent kinetic energy $k$ was set to 0 resulting in that the eddy dissipation $\epsilon$ also would need to be 0. For the turbulence models using $\omega$ this would mean that, when Eq. 3.20 holds, $\omega$ would be infinitely large. This is implemented just by choosing a very large value for $\omega$ at the boundary.

The just above discussed boundary conditions will demand that the first grid cells lay very near, let’s say $y^+ < 1$, to the wall. This would mean a lot of cells are needed to obtain a stable solution. If one knows how to model the near wall layer like done when one assumes a stationary turbulent boundary layer exists, the need for very fine grid in the near wall region is avoided.

5.2.5 Roughness parameter $E$

Solely looking at the results of the BAGT experiments the roughness is determined for the simulations of the SSB cases.

It seems like the smooth boundary will result in velocity profiles corresponding best with the experimental determined data. However one expects that the boundary should have an roughness and even more that an extra roughness effect should be present. This extra roughness is introduced to compensate for the momentum change due the acceleration of the eroded sand.

These two considerations lead to the choice, despite the lack of any proof, to continue with the roughness parameter for the transitional rough regime $E = 1.063$. 

CHAPTER 5. SIMULATING THE FLOW IN THE SCOUR HOLE

5.3 SSB

Now that the choice which surface roughness\(^1\) should be used is made, several cases from the SSB experiments are simulated.

Ideally all cases would be run, however that would have taken too much time as one case takes about 2 weeks of calculation time\(^2\). To get more insight, like what influence the nozzle diameter has for different inlet velocities, it is recommended after this work to simulate and evaluate more cases.

Deleted

The results of the individual simulations can be found in Appendix ???. In Appendix ?? also Figures to compare one on each other are given. The evaluation of the simulation results is done in the next section.

5.4 Evaluating the jet model results

In this chapter the simulation results are evaluated, the velocities and their profiles are first considered. Next the erosion velocities are investigated and possible relations are sought. The velocity profiles are plotted along the surface normal vector (inward pointed) on several locations along the chords. These chords are already defined in Section 5.4, bottom is the deepest point along the \(x\)-direction and the side the widest point along \(x\)-direction. In Figure 5.3 these profiles and their locations are schematically drawn. The starting points on of the normal vectors are chosen for several \(x\)-coordinates positions. An algorithm is used to calculate the distance of this point on the chords. The lines chosen to be plotted are given in table found in one of the appendices ?? After that the kinematic erosion velocity is compared with the erosion theory, and the shear velocities are considered.

5.4.1 Velocities

Deleted

\(^1\) Also an alternate boundary condition was used to implement the roughness: nutkRoughWallFunction which is given in Section 3.8.2.2. However using standard values for it resulted in an early separation of the flow and therefore not further investigated. Also one should look if and how the \(E\) parameter for the other wall functions is modified.

\(^2\) Based on the available hardware for this project.
Figure 5.3: Chords, blue the bottom chord and red the side chord, where the bottom is defined as the deepest point and side as the widest point for each position in $x$-direction.
5.4.2 Erosion

For the BAGT cases, the erosion velocities predicted by the erosion theory are lower than the values of the models like seen in Figure ???. However near the inlet good results are observed. To create a better feeling how the kinematic and calculated erosion correlate Figure 5.4a is given which show that the erosion theory only holds for the lower erosion velocities.

Deleted

Experiment 6 actually is showing some strange relations between the shear velocity and erosion velocity for the bottom. The asymptotic behaviour for the shear velocity versus the distance down stream is about the same order of magnitude as the first set. Suspected is that the more complex geometry is the cause of this.

To create a better feeling how the kinematic and calculated erosion correlate Figure 5.4b is given which show that the erosion theory only holds for the lower erosion velocities.

The erosion velocities for the SSB predicted by the erosion theory shown in Figure ?? are actually close to the measured (kinematic) erosion velocities (see Figure ??). The cases 1, 2 and 3 show good resemblance. Only at region near the nozzle, where the higher shear velocities occur, the kinematic erosion is higher than the calculated erosion. To create a better feeling how the kinematic and calculated erosion correlate Figure 5.4c is given, which show that the erosion theory only holds for the lower erosion velocities.

Deleted

5.4.3 Shear velocity

Deleted
CHAPTER 5. SIMULATING THE FLOW IN THE SCOUR HOLE

Figure 5.4: Simulation results kinematic erosion velocity plotted versus the calculated erosion velocity (a) BAGT, (b) SSB set 1 and (c) SSB set 2.
CHAPTER 5. SIMULATING THE FLOW IN THE SCOUR HOLE

\[ u = [m = s] \text{ from simulation} \]

Figure 5.5: Simulation results shear velocity plotted versus the calculated (a) BAGT, (b) SSB set 1 and (c) SSB set 2.
5.5 Conclusions

Deleted
Chapter 6

Conclusions and Recommendations

6.1 Conclusions

- The velocities in the scour hole determined by the CFD model are presented in this report, despite the assumptions and lack of validations these velocities are believed to be realistic.
- The obtained shear velocities result in almost identical erosion velocities compared to the kinematic erosion velocities for the bottom, at the side these calculated erosion velocities are underestimated. The region near the nozzle however the kinematic erosion velocity is higher than predicted by the erosion theory.

6.2 Recommendations

- Implement the k-ωSST in driftFluxFoam.
- Create one or several multiphase driftfluxfoam simulation(s) however a method for the mass injection should be thought of.
- Look for a relation which captures the entrainment of surrounding water.
- Investigate if a boundary layer for an eroding front exists and propose a model which capture the flow characteristics (like TBL layers). This would enable low calculation cost multiphase simulations.
- Validate current model build up using an static scour hole, for example in concrete, and measure at least the velocities and if possible determine the fluctuations. However measuring the turbulent boundary layer itself would be a big challenge.
CHAPTER 6. CONCLUSIONS AND RECOMMENDATIONS

- Use the similarity aspects presented in this thesis to update the current jet model.
- Use the found relations between the flow and shear velocity presented in this thesis model to a growth of the hole. Hopefully a stationary scour hole is obtained after some time.
- Sand injection could be introduced with a kinematic boundary condition. But because of the lack of knowledge the boundary layers (if exist) should be solved numerically. One could do this using \( k = 0, u = 0 \) and \( \omega = \infty \) (or \( \epsilon = 0 \)). However computer costs will be too much for nowadays, because it is expected that a very fine grid is needed in order to obtain correct results.
Literature


A. Shields. Application of similarity principles and turbulence research to bed-load movement. Hydrodynamics Laboratory, 1(167), 1936. Trans. from Anwendund der Aehnlichkeitsmechanik und der Turublenzforschung auf die Geschiebe bewegung.


Ronald van der Hout. Validation of the physical jet models on the measured 3d jet bodies in sand. Confidential, internal use only, Jan 2010.


Appendix A

Two-equation models

Two-equation models generally uses equations to model not only the turbulent viscosity but also the turbulence characteristics. Therefore a solution can be found without prior knowledge of the turbulence in the flow like needed in the one-equation models, also these models are complete. In this thesis three two-equation turbulence models are discussed, k-\(\epsilon\), k-\(\omega\) and k-\(\omega_sST\). The latter is actually a blend of the first two.

A.1 The k-\(\epsilon\) model

The equation describing the turbulent kinetic energy is given by Eq. 3.31. To obtain a model \(\epsilon\) equation (which describes the dissipation of the turbulent kinetic energy) the following terms of Eq. 3.27 are simplified the motivation can be found in (Celik, 1999):

I.\(\text{II}\) assuming a local equilibrium and the rate at which \(k\) is produced must be equal to \(\epsilon\) and therefore must scale with \(k\) which gives \(C_{\epsilon 1}\)

III based on dimensional analysis the destruction of dissipation results in \(C_{\epsilon 2}\)

IV The transport is modelled by applying a well known gradient diffusion, only now using an other turbulent Prandtl number for the dissipation resulting in

\[
\frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_{\epsilon}} \frac{\partial \epsilon}{\partial x_j} \right]
\]

So the dissipation rate (in differential form) for the k-\(\epsilon\) model becomes:

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

(A.1)

and

\[
\mu_t = \frac{\rho C_{\mu} k^2}{\epsilon}
\]

(A.2)

\(C_{\epsilon 1} = 1.44, \quad C_{\epsilon 2} = 1.92, \quad C_{\mu} = 0.09, \quad \sigma_k = 1.0, \quad \sigma_{\epsilon} = 1.3\)
A.2 The $k$-$\omega$ model

The rate of dissipation $\epsilon$ is not the only scale which turbulence is tried to be modelled with, looking at the turbulence frequency $\omega$ another model could be introduced. Kolmogrov\(^1\) (using $\omega \sim k^{1/2}/\ell$) introduced:

$$\rho \frac{\partial \omega}{\partial t} + \rho U_j \frac{\partial \omega}{\partial x_j} = -\beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ \sigma \mu_t \frac{\partial \omega}{\partial x_j} \right]$$  \hspace{1cm} (A.3)

At the wall omega tends to go to infinity as the length of the eddies goes to zero at the wall. It was later realized that the absence of a relation with the kinematic energy $k$ the model was unreliable. There for the following equations where introduced:

$$\rho \frac{\partial \omega}{\partial t} + \rho U_j \frac{\partial \omega}{\partial x_j} = \alpha^* \omega \frac{\tau_{ij}}{k} \frac{\partial U_i}{\partial x_j} - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ \sigma \mu_t \frac{\partial \omega}{\partial x_j} \right]$$  \hspace{1cm} (A.4)

The kinetic energy equation rewritten for using $\omega$ reads:

$$\rho \frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta^* \rho_k \omega^2 + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma^* \mu_t) \frac{\partial k}{\partial x_j} \right]$$  \hspace{1cm} (A.5)

The eddy viscosity and the closure constants can be expressed as:

$$\mu_t = \frac{\rho k}{\omega}$$

$$\alpha^* = 5/9, \hspace{0.5cm} \beta = 3/40, \hspace{0.5cm} \beta^* = 9/100, \hspace{0.5cm} \sigma = 1/2, \hspace{0.5cm} \sigma^* = 1/2$$

This model has a rather physical interpretation to apply for rough walls: in addition to the standard wall treatment using some empirical constants an definition using an equivalent sand grain diameter is used as can be seen later in Section 3.6.3.

A.3 The $k$-$\omega$ SST model

OpenFOAM also provides the turbulence model introduced by Menter: $k$-$\omega$ SST where the last term stands for Shear Stress Transport, it provides a mix of the $k$-$\epsilon$ (any where outside the boundary layers, to avoid dependence of the free stream conditions) and the Wilcox $k$-$\omega$ model for inside the boundary layer (i.e. near the wall) (Menter, 1994). Menter formulated a so-called baseline model (BSL) and (Hellsten, 1997) provided an improvement introducing a blending for rough walls, the model is given next.

The $k$-$\epsilon$ model is rewritten in a form using $\omega$ by using Eq. 3.20 resulting in:

$$\rho \frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial x_j} = P_k - \beta^* \rho_k \omega + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$  \hspace{1cm} (A.7)

\(^1\)Kolmogrov also stated that dissipation occurs at the smallest eddy scale thus the rate of dissipation is the rate of turbulent kinetic energy transfer from the larger to the smaller eddies.
APPENDIX A. TWO-EQUATION MODELS

\[ \frac{\partial \omega}{\partial t} + \rho U_j \frac{\partial \omega}{\partial x_j} = \frac{\gamma \rho}{\mu_t} P - F_4 \beta^* \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + 2\rho \frac{1 - F_1}{\sigma_\omega^2} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \] (A.8)

In the last equation \( F_4 \) regulates the rotation and curvature sensitization however this is not implemented in the solver of OpenFOAM i.e. it should be thought of that it is equal to unity. The model closure coefficients are also ‘blended’ and can be found by using original Wilcox (denoted by subscript 1) and the transformed standard \( k-\epsilon \) (denoted by subscript 2) coefficients. The relation to determine the new is:

\[
\begin{pmatrix}
\sigma_k \\
\sigma_\omega \\
\beta
\end{pmatrix} = F_1 \begin{pmatrix}
\sigma_{k1} \\
\sigma_{\omega 1} \\
\beta_1
\end{pmatrix} + (1 - F_1) \begin{pmatrix}
\sigma_{k2} \\
\sigma_{\omega 2} \\
\beta_2
\end{pmatrix}
\] (A.9)

And the closure coefficients:

\[
\kappa = 0.41, \quad \beta^* = 0.09
\]
\[
\sigma_{k1} = 0.85, \quad \sigma_{\omega 1} = 0.5, \quad \beta_1 = 0.075
\] (A.10)
\[
\sigma_{k2} = 1.0, \quad \sigma_{\omega 2} = 1.168, \quad \beta_2 = 0.0828
\]

Note that for the original (BSL) model that \( \sigma_{k1} = 2.0 \) and \( \sigma_{\omega 1} = 2.0 \). The constant \( \gamma \) can be calculated using

\[
\gamma = \frac{\beta}{\beta^*} - \frac{\sigma_\omega \kappa^2}{\sqrt{\beta^*}}
\] (A.11)

The production of turbulent kinetic energy \( P_k \) is modelled using Boussinesq assumption analogue to Eq. 3.28:

\[
P_k = -\rho u_i u_j \frac{\partial u_i}{\partial x_j} = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j}
\] (A.12)

The blending function \( F_1 \) is given by:

\[
F_1 = \tanh \left( \Gamma_1^4 \right)
\] (A.13)

where

\[
\Gamma_1 = \min \left( \max \left( \sqrt{\frac{k}{\beta^* \omega n_w}}, \frac{500\nu}{\omega n_w}, \frac{4\rho \sigma_\omega k}{CD_{k\omega} n_w}, \frac{2\rho \partial k}{\sigma_\omega^2 \omega \partial x_j \partial x_j} \right) \right)
\] (A.14)

Again \( n_w \) denotes the (shortest) distance to the surface and the cross diffusion term \( CD_{k\omega} \) is defined as:

\[
CD_{k\omega} = \max \left( \frac{2\rho \partial k}{\sigma_\omega^2 \omega \partial x_j \partial x_j}, \frac{10^{-20}}{1} \right)
\] (A.15)

The term \( I \) in Eq. A.14 tends to 2.5 in the log layer and becomes zero in the defect layer. The term \( II \) exceeds unity in the sublayer and the third term \( III \) ensures correct behavior of \( F_1 \) for free streams. The (dynamic) eddy viscosity is given by

\[
\mu_t = \frac{a_1 \rho k}{\max (a_1 \omega, |S_{ij}| F_2 F_3)}
\] (A.16)
APPENDIX A. TWO-EQUATION MODELS

The constant $a_1$ should be chosen as $a_1 = 0.31$ and $|S_{ij}| = \sqrt{S_{ij}^2}$ and the strain-rate tensor terms can be calculated using Eq. 3.30. The second blending function $F_2$ is given by (it ensures that limiting the viscosity for free shear flows is not activated for shear flows):

$$F_2 = \tanh(\Gamma_2^2)$$ (A.17)

where

$$\Gamma_2 = \max \left( \frac{2\sqrt{k}}{\beta^*\omega n}, \frac{500\nu}{\omega n^2} \right)$$ (A.18)

The function $F_3$ is introduced to prevent, like $F_2$, activation of the limitation of the viscosity in the rough wall layer and is given by:

$$F_3 = 1 - \tanh \left( \frac{150\nu}{\omega n^2} \right)^4$$ (A.19)
Appendix B

Drift flux model

The drift flux model looking at as the dispersed phases an continuous phase as a mixture, so only one continuous phase is considered. The work of (Goeree et al., 2013) is discussed here.

B.1 Mixture

The density of the mixture can be determined from:

\[ \rho_m = \sum_{k=1}^{N} \rho_k \alpha_k \]  

(B.1)

The sub index \( k \) denotes the number of each phase and \( N \) is the total number of phases in the mixture, \( \alpha \) symbol represents the volume averaged quantity of phase \( k \). An useful definition is the total volume concentration:

\[ \sum_{k=1}^{N} \alpha_k = 1 \]  

(B.2)

And the solids volume concentration is logically defined as:

\[ \alpha_t = \sum_{k=1}^{N} \alpha_k - \alpha_f \]  

(B.3)

The fraction \( \alpha_f \) denotes the liquid phase volume concentration (clear water). In the drift flux model the mixture velocity is obtained by mass weighted averaging:

\[ \mathbf{u}_m = \sum_{k=1}^{N} c_k \mathbf{u}_k \]  

(B.4)

In this \( c_k \) is the mass fraction of phase \( k \) and \( \mathbf{u}_k \) is its velocity. The phase mass fraction can be determined using:

\[ c_k = \frac{\rho_k \alpha_k}{\rho_m} \]  

(B.5)
APPENDIX B. DRIFT FLUX MODEL

Now the dispersed phase (or particles) will most likely move with different velocity with respect to the liquid phase and also with respect to the mixture velocity. The relative velocities are defined by the following, respectively the:

\[ \mathbf{u}_{k} = \mathbf{u}_{k} - \mathbf{u}_{f} \]  
(B.6)

and

\[ \mathbf{u}_{km} = \mathbf{u}_{k} - \mathbf{u}_{m} \]  
(B.7)

The liquid phase velocity is denoted by \( \mathbf{u}_{f} \). Also the relative velocity between the dispersed and liquid phase is \( \mathbf{u}_{kf} \) and the relative velocity between the dispersed and liquid phase is \( \mathbf{u}_{km} \). In the report of Goeree it is given that the relative velocity \( \mathbf{u}_{kf} \) is calculated by applying a closure relation such as the relation of Richardson and Zaki.

If it is assumed that each phase \( k \) has its own momentum equation:

\[
\frac{\partial \rho_{k} \mathbf{u}_{k}}{\partial t} + \nabla \cdot (\rho_{k} \mathbf{u}_{k} \mathbf{u}_{k}) = -\nabla p_{k} + \nabla \cdot (\tau_{k} + \tau_{k}^{\text{turb}}) + \rho_{k} \mathbf{g} + \mathbf{M}_{k} \]  
(B.8)

The stress viscous and turbulent stresses are denoted by \( \tau_{k} \) and \( \tau_{k}^{\text{turb}} \). The interaction of each fraction is captured in the source term \( \mathbf{M}_{k} \) and the body forces like gravitation is represented by \( \mathbf{g} \). And finally the pressure of each fraction is \( p_{k} \). One can derive the mixture momentum equation by summing over all volume fractions and use the definitions in this section. The equation obtained is:

\[
\frac{\partial \rho_{m} \mathbf{u}_{m}}{\partial t} + \nabla \cdot (\rho_{m} \mathbf{u}_{m} \mathbf{u}_{m}) = -\nabla p_{m} + \nabla \cdot (\tau_{m} + \tau_{m}^{\text{turb}} - \sum_{k=1}^{N} \alpha_{k} \rho_{k} \mathbf{u}_{kf} \mathbf{u}_{kf}) + \rho_{m} \mathbf{g} \]  
(B.9)

In the above equation the mixture interaction (not shown here) force can be removed because all forces are opposite to another (Goeree et al., 2013). And for the incompressible case the mixture continuity equation:

\[
\frac{\partial \rho_{m}}{\partial t} + \nabla \cdot (\rho_{m} \mathbf{u}_{m}) = 0 \]  
(B.10)

By looking at the sediment transport we can find the \( \mathbf{u}_{kf} \) term as follows. For a sediment phase transport the continuity equation, where it is obviously that no mass can be destroyed nor created, is:

\[
\frac{\partial \rho_{k}}{\partial t} + \nabla \cdot (\rho_{k} \alpha_{k} \mathbf{u}_{k}) = 0 \]  
(B.11)

The phase density \( \rho_{k} \) is constant so could be omitted. The phase velocity can be calculated using earlier equations combined:

\[
\mathbf{u}_{k} = \mathbf{u}_{km} + \sum_{k=1}^{N} c_{k} \mathbf{u}_{k} \]  
(B.12)

where

\[
\mathbf{u}_{kf} = \mathbf{u}_{kf} - \sum_{k=1}^{N} c_{k} \mathbf{u}_{kf} \]  
(B.13)
The closure relation introduced for hindered settling:

\[ u_{kf} = u_{k\infty}V_k(\alpha_t) \]  \hspace{1cm} (B.14)

In this the term \( u_{k\infty} \) is the terminal settling velocity and the hindered settlement function is \( V_k(\alpha_t) \), which (Goeree et al., 2013) suggest to be from Basson, Mirza and Richardson:

\[
V_k(\alpha) = (1 - \alpha_t)^{n_k-1} \quad \text{for } \alpha_t < \alpha_{t,\text{max}} \hspace{1cm} (B.15a)
\]

\[
V_k(\alpha) = 0 \quad \text{for } \alpha_t < \alpha_{t,\text{max}} \hspace{1cm} (B.15b)
\]

and where index \( n_k \) can be calculated by:

\[
n_k = 2.35 \frac{2 + 0.175Re_p^{3/4}}{1 + 0.175Re_p^{3/4}} \hspace{1cm} (B.16)
\]
Appendix C

Plane Wall Jet

C.0.1 Equations

From Rajaratnam the simplified equations of motion (no laminar shear stress and ignoring small terms compared to the larger terms (velocity fluctuations gradient in x-direction)) and assuming a small pressure gradient in x-direction:

\[ U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} = \frac{1}{\rho} \frac{\partial \tau_{tl}}{\partial y} \]  
\[ \frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} \]  

(C.1)

The term \( \tau_{tl} \) is the turbulent and laminar shear stress \( \rho u'v' + \mu \frac{\partial U}{\partial y} \). Acknowledging that in this simplifications the momentum should be preserved, a momentum balance will be can be obtained. Multiplying Eq. C.1 by \( \rho \) one obtains the momentum and consecutively one can integrate along \( y \) to obtain the momentum flux in \( x \)-direction:

\[ \rho \int_{0}^{\infty} U \frac{\partial U}{\partial x} dy + \rho \int_{0}^{\infty} V \frac{\partial U}{\partial y} dy = \int_{0}^{\infty} \frac{\partial \tau_{tl}}{\partial y} dy \]  

(C.3)

which can be rewritten (Rajaratnam, 1976):

\[ \frac{d}{dx} \int_{0}^{\infty} \rho U^2 dy = -\tau_w \]  

(C.4)

The wall shear stress is denoted as \( \tau_w \). From looking at Eq. C.4 considerations one obtains (rewrite first in terms of \( f(\eta) \)):

\[ U_m \propto 1/\sqrt{x}, \quad b \propto x, \quad \tau_w \propto 1/x \]  

(C.5)

further using dimensional considerations one gets:

\[ \frac{U_m}{U_0} = C_1 / \sqrt{x/b_0} \]  
\[ b = C_2 x \]  

(C.6)  

(C.7)
\[ \tau_w = C_3 \frac{\rho U_0^2}{2(x/b_0)} \]  \hspace{1cm} (C.8)

The coefficients \( C_1, C_2 \) and \( C_3 \) are to be determined, if \( \nu \) would have also been considered to determine the above mentioned relations than these coefficients would vary with the nozzle Reynolds number, however (Rajaratnam, 1976) states that experimental studies indicate that this variation can be neglected\(^1\).

### C.0.2 Found relations

The similarity of the wall jet is determined by Verhoff and can be found in (Rajaratnam, 1976)

\[ \frac{U}{U_m} = 1.48\eta^{1/7} [1 - \text{erf}(0.68\eta)] \]  \hspace{1cm} (C.9)

in this \( \eta \) is the non-dimensional height defined as \( \eta = z/b \) and \( \text{erf} \) is the so-called error function.

The coefficient \( C_1 \) is taken in (Rajaratnam, 1976) as 3.50 and invariant of the nozzle Reynolds number. However looking at Figure ?? one might as well choose \( C_1 \) equal to 3.65. If using the second given value and the fact that the maximum velocity \( u_m \) anywhere can not exceed the inlet velocity \( U_0 \) (if the profile is assumed to be developed) the velocity scale of the jet can be described by:

\[ \frac{U_m}{U_0} = \min \left( 1; \frac{3.65}{\sqrt{x/b_0}} \right) \quad \text{for} \quad \frac{x}{b_0} \leq 100 \]  \hspace{1cm} (C.10)

Note that Eq. C.10 can never result in negative values. Another possibility is using a blending of the developed and undeveloped region, and does not show a sharp point of decay:

\[ \frac{U_m}{U_0} = C_1 \frac{\tanh \left( \frac{x/b_0}{a_2} \right)^p}{2} + 1 \cdot \left( 1 - \tanh \left( \frac{x/b_0}{a_2} \right)^{p_2 - 1} \right) \]  \hspace{1cm} (C.11)

Constants \( a_2 \) and \( p_2 \) are found by curve fitting on the experimental data shown in Figure C.1 together with the other functions the blended function is plotted with the constants \( a_2 = 9.734 \) and \( p_2 = 6 \).

As mentioned earlier it is believed that \( C_1 \) depends on the nozzle Reynolds number \( Re_0 = U_0b_0/\nu \), so one gets \(^2\):

\[ C_1^* \approx \frac{1}{\sqrt{0.1 - 1.25Re_0 \cdot 10^{-6}}} \]  \hspace{1cm} (C.12)

However the limit of this function as \( Re_0 \to \infty \) will result in negative roots, therefore it should not be used at these high Reynolds numbers (\( Re_0 \approx 80000 \)).

---

\(^1\)Later in this report the Reynolds independence assumption can be considered to be questionable

\(^2\)It is good to realise that this is not a physical relation and is neither based on dimensional considerations! It is solely fitted to experimental data which is (rather primitive) presented in (Rajaratnam, 1976)!
Figure C.1: Velocity scales for plane wall jets digitised from (Rajaratnam, 1976), added Eq. C.6 for $C_1 = 3.65$, the blended function Eq. C.11 with $a_2 = 9.734$ and $p_2 = 6$ and a limited function Eq. C.10

Just leaving Eq. ?? for what it is we get back to Eq. C.10 and Eq. C.11, these functions are plotted in Figure C.1.

The constant $C_2$ in Eq. ?? used for velocity scale growth relation is determined to be equal to 0.068 so Eq.?? becomes:

$$b = 0.068x$$ \hspace{1cm} (C.13)

The wall shear stress is rewritten, see for example (Rajaratnam, 1976), in two different forms and the empirically found relations also given. The results are two different skin friction factors, Myers reported:

$$c_1 \equiv \frac{\tau_w}{\rho U_0^2/2} \simeq \frac{0.20}{x/b_0(U_0b_0/\nu)^{1/12}} = \frac{0.20}{x/b_0(R_0)^{1/12}} = 0.20$$ \hspace{1cm} (C.14)

and Sigalla:

$$c_1' \equiv \frac{\tau_w}{\rho U_m^2/2} = \frac{0.0565}{(U_m y_m/\nu)^{1/4}} = 0.0565$$ \hspace{1cm} (C.15)

Note in 2.13 the nozzle Reynolds number is present. In Eq. 2.14 the $y_m$ represents the distance from the point where the velocity in the jet is maximal to the (nearest) surface as defined in Figure ??, most literature calls this the local Reynolds number defined as:

$$Re_m = U_m y_m / \nu$$ \hspace{1cm} (C.16)

Reliable data on skin friction is reported by Eriksson (see Section 4.3.1) and is plotted together with Sigalla’s relation in Figure C.2.
APPENDIX C. PLANE WALL JET

Figure C.2: The skin friction coefficient versus the local Reynolds number, experimental results from Erikkson and Abrahamson together with rough data found in the report of Rostamy see Section 4.3.2. Also the empirical found relation from Sigalla Eq. C.15 and Erikkson Eq. 4.2.

The entrainment velocity is expressed in (Rajaratnam, 1976) as:

\[ v_\alpha = \frac{dQ}{dx} = \alpha_e U_m = 0.035 U_m \tag{C.17} \]

The coefficient \( \alpha_e = 0.035 \) is the entrainment coefficient.

Rough walls also have an influence on the velocity scales

\[ \frac{U_m}{U_0} = \max \left( \min \left( 1; C_R - 0.54 \log_{10} \left( \frac{x}{b_0 \varepsilon_{b_0}} \right) \right); 0 \right) \tag{C.18} \]

The parameter \( \varepsilon_{b_0} \) represents the effective roughness relative to the nozzle:

\[ \varepsilon_{b_0} = \frac{k_e}{b_0} \tag{C.19} \]

The coefficient \( C_R \) is dependent on the relative roughness \( \varepsilon_{b_0} \) and \( k_e \) denotes the effective (absolute) roughness. The \( \max() \) function is added by the writer to Eq. C.18 because it should be prevented that negative maximum velocities occur, the \( \min() \) function is also added for the same reason as for the smooth velocity scale function. The coefficient \( C_R \) is vaguely presented in (Rajaratnam, 1976): only as a curve with some experimental found values where parameters like nozzle Reynolds numbers are not given. This figure is again digitalised and shown in Figure C.3. In order to have an expression to predict
Figure C.3: Variation of the velocity-decay coefficient with relative roughness digitised from (Rajaratnam, 1976) and a curve fitted relation Eq. C.20

The plane yet on rough surfaces a exponential form is assumed by the writer of this report

\[ C_R = e^{-16.85 \frac{k_e}{b_0}} + 1.664 \]  \hspace{1cm} (C.20)

Equation C.18 is shown in Figure C.4, together with digitised measurements from (Rajaratnam, 1976) and the 'smooth' wall Eq. C.10.

\[^3\text{This is solely based on curve fitting and not based on any physical or dimensional considerations!} \]

95
Figure C.4: Roughness effects on the velocity scales for plane rough wall jets digitised from (Rajaratnam, 1976), added Eq. C.6 for $C_1 = 3.65$ representing the smooth wall, and Eq. C.18 representing rough wall with roughness $k_e/b_0$
Appendix D

Plane Wall Jet Grid Generation

The grid is build using the OpenFOAM blockMesh (see (Unknown, 2014)) utility defining two hexahedrons, one for the near wall region (with an height equal to \( b_0 \)) and one for the large region above the first. The nonuniform mesh grading is done by setting the ratio of the first and last cell \( R \). From (Unknown, 2014) one finds that the smallest cell \( \delta x_s \) can be determined by:

\[
\delta x_s = b_0 \frac{r - 1}{\alpha r - 1} \tag{D.1}
\]

where the height of our near wall hexahedron \( b_0 \) is already used as the reference length. \( r \) is the ratio of one cell \( i \) size and its next cell \( i + 1 \) and is given by:

\[
r = R^{\frac{i}{i-1}} \tag{D.2}
\]

and \( \alpha_r \) is:

\[
\alpha_r = R \quad \text{for} \quad R > 1 \tag{D.3a}
\]

\[
\alpha_r = 1 - r^{-i} + r^{-1} \quad \text{for} \quad R < 1 \tag{D.3b}
\]
Appendix E

Open Channel

E.1 Introduction

As reference experiments the Coleman experiments are used and taken from Winterwerp. Chosen is a height of $h = 0.172m$ and an inlet (uniform of $u = 1m/s$) and the total length of the channel is $8m$.

E.2 Simulation

The domain (2D) is build upon $800 \times 17$ cells, and an simple grade of 2 in both directions. The boundary conditions of each relevant parameter are shown in table E.1.

The results are shown in figure E.1, where the dimensionless velocity (the ratio of the velocity $u$ over the average cross sectional velocity $U_{avg}$) is plotted over the dimensionless height (the ratio of the height from the bottom surface $z$ over the total height of the channel $h$).

E.3 Conclusion

Although similar boundary conditions are used the two different turbulence models produces almost similar results. However differences are present
Table E.1: The boundary conditions applying for the open channel simulation using the $k - \epsilon$ or $k - \omega_{SST}$ turbulence model

<table>
<thead>
<tr>
<th>Patch</th>
<th>Field</th>
<th>Type</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>inlet</td>
<td>$U$</td>
<td>fixedValue</td>
<td>inlet profile</td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>turbulentIntensityKineticEnergyInlet</td>
<td>intensity 1</td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>calculated</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\epsilon$</td>
<td>turbulentMixingLengthDissipationRateInlet</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>fixedValue</td>
<td>uniform $1e - 15$</td>
</tr>
<tr>
<td>outlet</td>
<td>$U$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>fixedValue (uniform 0)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>calculated</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\epsilon$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td>bottom</td>
<td>$U$</td>
<td>fixedValue</td>
<td>uniform $(0 0 0)$</td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>nutUSpaldingWallFunction</td>
<td></td>
</tr>
<tr>
<td>top</td>
<td>$U$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>fixedValue</td>
<td>uniform 0</td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>calculated</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\epsilon$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\omega$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td>other</td>
<td>$U$</td>
<td>fixedValue</td>
<td>uniform $(0 0 0)$</td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>zeroGradient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\nu_t$</td>
<td>nutUSpaldingWallFunction</td>
<td></td>
</tr>
<tr>
<td>front</td>
<td>all</td>
<td>empty</td>
<td></td>
</tr>
<tr>
<td>back</td>
<td>all</td>
<td>empty</td>
<td></td>
</tr>
</tbody>
</table>
Figure E.1: Results of the open channel simulation at $x = 8m$, two turbulence models are compared with Coleman experiments.
Appendix F

Approximation of shear stress influence

F.1 In pipe

Let us consider a pipe flow where the exact volume of water is extracted as the volume of sand is injected as shown in Figure F.1.

Conservation of volume:

\[ u_1 h_1 + v_c L = u_2 h_2 + v_c L \]  \hspace{1cm} (F.1)

Conservation of mass:

\[ u_1 h_1 \rho_1 + v_c c_b \rho_s + v_c (1 - c_b) L \rho_w = u_2 h_2 \rho_2 + v_c c_b \rho_s + v_c (1 - c_b) L \rho_w \]  \hspace{1cm} (F.2)

Conservation of momentum along flow direction:

\[ u_1^2 h_1 \rho_1 = u_2^2 h_2 \rho_2 + \tau_{eq} \]  \hspace{1cm} (F.3)

In the last equation the \( \tau_{eq} \) is the unknown equivalent shear stress. For the pipe

\[ h_1 = h_2 = h \]  \hspace{1cm} (F.4)

Substituting in Eq. F.4 in F.1 and simplifying will result in:

\[ u_1 = u_2 = u \]  \hspace{1cm} (F.5)

And using again Eq. F.4 and the new Eq. F.5 one gets:

\[ u h (\rho_1 - \rho_2) = v_c c_b (\rho_w - \rho_s) \]  \hspace{1cm} (F.6)

And using again Eq. F.4 and the new Eq. F.5 by substituting in F.3 one gets:

\[ u (u h (\rho_1 - \rho_2)) = \tau_{eq} \]  \hspace{1cm} (F.7)

Finally substituting F.6 in F.7:

\[ u (v_c L c_b (\rho_w - \rho_s)) = \tau_{eq} \]  \hspace{1cm} (F.8)

Note that the shear stress would always be positive meaning that it ‘acts’ as a accelerator to maintain \( u_1 = u_2 \)
APPENDIX F. APPROXIMATION OF SHEAR STRESS INFLUENCE

Figure F.1: Principal drawing used to estimate the extra shear stress due to mass injection in a pipe
Appendix G

Permeability

In order to use the erosion formulae the permeability of the dilated layer $\lambda_l$ should be determined. This is approximated using the following relations found in Verruijt (2012).

$$\lambda = \frac{\kappa \lambda g}{\nu} \quad (G.1)$$

In this, $\kappa_\lambda$ denotes the intrinsic permeability, it can be determined using Kozeny-Carman:

$$\kappa_\lambda = C_\kappa d^2 \frac{n^3}{(1-n)^2} \quad (G.2)$$

In which $n$ is the porosity $C_\kappa$ is a constant, $d$ the particle size diameter.

So we need first to determine the $C_\kappa$ constant, and this is possible using the available permeability data from SSB (averaged):

$$n_0 = -$$ \quad (G.3a)
$$\lambda_0 = -$$ \quad (G.3b)

So

$$\kappa_{\lambda,0} = -$$ \quad (G.4a)
$$C_\kappa = -$$ \quad (G.4b)

Now calculating the permeability for $n_l$ is possible assuming:

$$n_l = -$$ \quad (G.5)

resulting in:

$$\kappa_{\lambda,l} = -$$ \quad (G.6a)
$$\lambda_l = -$$ \quad (G.6b)
Appendix H

Chronologic overview
Figure H.1: Summary of project outline