Flow through and around fish farming nets

Oystein Patursson
University of New Hampshire, Durham

Follow this and additional works at: https://scholars.unh.edu/dissertation

Recommended Citation
https://scholars.unh.edu/dissertation/434

This Dissertation is brought to you for free and open access by the Student Scholarship at University of New Hampshire Scholars' Repository. It has been accepted for inclusion in Doctoral Dissertations by an authorized administrator of University of New Hampshire Scholars' Repository. For more information, please contact nicole.hentz@unh.edu.
Flow through and around fish farming nets

Abstract
Computational fluid dynamics (CFD) modeling, tow tank and field measurements were used to investigate current flow through and around net panels and cages. For the numerical computations a porous media model was used to represent the net allowing efficient computation of both exterior and interior flow fields. The model was calibrated using tow tank measurements on a net panel at different velocities and angles of attack. The CFD method was able to reproduce the drag- and lift coefficients of the net panel and the velocity reduction behind the net panel with satisfactory accuracy.

The approach was validated for a small size gravity cage by comparing CFD predictions with tow tank measurements of drag force on the cage and velocity reduction inside the cage and in the wake region. The modeled drag force was higher than the measured drag force. The modeled current compared well with the measured current inside the cage, but the reduction was underpredicted in the wake of the cage.

Full scale simulations were performed for a cage with a clean net and a biofouled net and compared with field measurements of a cage fouled with jellyfish. The measured data compared well with model predictions for the biofouled net. Flushing rates were calculated for both the clean and the biofouled net cases. When the net was changed from clean to biofouled, flushing time increased by up to 44% and drag force increased by up to 80%.

Keywords
Engineering, Marine and Ocean, Agriculture, Fisheries and Aquaculture

This dissertation is available at University of New Hampshire Scholars' Repository: https://scholars.unh.edu/dissertation/434
FLOW THROUGH AND AROUND FISH FARMING NETS

BY

Øystein Patursson
B.S., University of the Faroe Islands, 2002

DISSERTATION

Submitted to the University of New Hampshire
in partial fulfillment of
the requirements for the degree of

Doctor of Philosophy
in
Ocean Engineering

May 2008
INFORMATION TO USERS

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleed-through, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

UMI Microform 3308384
Copyright 2008 by ProQuest LLC.
All rights reserved. This microform edition is protected against unauthorized copying under Title 17, United States Code.

ProQuest LLC
789 E. Eisenhower Parkway
PO Box 1346
Ann Arbor, MI 48106-1346
This dissertation has been examined and approved.

M. Robinson Swift
Dissertation Director, M. Robinson Swift
Professor of Mechanical and Ocean Engineering

Kenneth Baldwin
Professor of Mechanical and Ocean Engineering

Barbaros Celikkoğlu
Professor of Mechanical and Ocean Engineering

David Fredriksson
Assistant Professor of Ocean Engineering,
U. S. Naval Academy

Knud Simonsen
Associate Professor of Oceanography,
University of the Faroe Islands

Igor Tsukrov
Associate Professor of Mechanical Engineering

04/02/08
Date
DEDICATION

To the memory of my father
ACKNOWLEDGMENTS

The present work is part of a PhD project organized as a collaborative project between the University of the Faroe Islands and the University of New Hampshire. Most of the funding is from the Statoil Group, which is a consortium of oil companies, including Statoil, Phillips, Enterprise, and Veba who are involved in oil exploration on the shelf of the Faroe Islands, however the project is also funded by the Cooperative Institute for New England Mariculture and Fisheries (CINEMAR) and the National Oceanic and Atmospheric Administration (NOAA).

The input from my supervisory committee, professors M. Robinson Swift, Kenneth Baldwin, Barbaros Celikkol and Igor Tsukrov from the University of New Hampshire, David Fredriksson from the US Naval Academy and Knud Simonsen from the University of the Faroe Islands, on the modeling approach, calibration and validation steps and on the design of the experiments is highly appreciated. I would also like to thank Mr. Paul Lavoie for his help with designing and making the setup for measuring forces on the net panel and Mr. Glen Rice for welding and putting in late hours with the setup and calibration. I also want to thank the U.S. Naval Academy Hydromechanics Laboratory staff for their help with the cage tow testing and for using their acoustic doppler velocimeter (ADV) for the net panel tests and Dr. James Irish and NOBSKA for using their MAVS-3 current meters.

I also want to thank my family for their support and patience when things were not progressing as planned and workdays were long.
TABLE OF CONTENTS

DEDICATION ........................................ iii
ACKNOWLEDGMENTS ................................. iv
LIST OF TABLES .................................. viii
LIST OF FIGURES ................................. x
ABSTRACT ......................................... xiv

1 INTRODUCTION ................................. 1
1.1 Cage Flow Problem .......................... 1
   1.1.1 Background ............................. 1
1.2 Previous Technical Work .................... 3
1.3 Goals ......................................... 7
1.4 Approach .................................... 8

2 THEORETICAL CONSIDERATIONS .......... 10
2.1 Hydrodynamic Equations Solved .......... 11
   2.1.1 Modeling Turbulence through Reynolds Averaging .... 12
   2.1.2 The Standard $k - \epsilon$ Model ............ 15
   2.1.3 The Realizable $k - \epsilon$ Model ........... 16
   2.1.4 The Standard $k - \omega$ Model ............ 18
   2.1.5 Flow through Porous Media .............. 20
2.2 Boundary Conditions ...................... 25
2.2.1 Velocity Inlet Boundary Condition ............................. 26
2.2.2 Pressure Outlet Boundary Condition ............................. 27
2.2.3 Wall Boundary Condition ........................................... 28
2.2.4 Near Wall Treatment ................................................ 28
2.3 Finite Volume Discretization .......................................... 30
  2.3.1 Discretization of the Scalar Transport Equation ................. 31
  2.3.2 Discretization Schemes ............................................ 32
  2.3.3 Evaluation of Derivatives ......................................... 33
  2.3.4 Solver .............................................................. 34
  2.3.5 Discretization of the Momentum Equation ......................... 35
  2.3.6 Discretization of the Continuity Equation ....................... 36
  2.3.7 Pressure Velocity Coupling ...................................... 37
  2.3.8 Algebraic Multigrid ............................................... 38
  2.3.9 Under Relaxation ................................................ 39
2.4 Meshing .............................................................. 40
  2.4.1 Grid Generating Methods ........................................... 40
  2.4.2 Numerical Diffusion ................................................ 43
  2.4.3 Refinement of the Mesh ............................................ 44
2.5 Using the Model ....................................................... 44
2.6 Control Volumes ....................................................... 46

3 NET PANEL STUDIES ......................................................... 50
  3.1 Measurements .......................................................... 52
    3.1.1 Measurement Setup ............................................... 52
3.1.2 Dataprocessing .............................................. 58
3.1.3 Results and Discussion ................................... 60

3.2 Preliminary Computational Fluid Dynamic Runs ........... 63
  3.2.1 Testing Dependence on Thickness of Porous Media ...... 63
  3.2.2 Testing Dependence on Models Used in FLUENT ........ 65
  3.2.3 CFD-Mesh used for Net Panel and Frame ................ 68
  3.2.4 Turbulence at the Inlet Boundary ....................... 71
  3.2.5 Conclusions from the Preliminary CFD Runs ........... 72

3.3 Porous Media Resistance Coefficients ....................... 74
  3.3.1 Analytical Approach for Calculating Forces on the Net Panel 74
  3.3.2 Finding the best Porous Media Resistance Coefficients ... 78

3.4 Results from Computational Fluid Dynamics ................. 86
  3.4.1 Accuracy of the Porous Resistance Coefficient Procedure .. 88

3.5 Discussion .................................................. 90

4 GRAVITY CAGE STUDIES ........................................... 92
  4.1 Tow Tank Experiments ...................................... 92
    4.1.1 Fish Cage Particulars .................................. 93
    4.1.2 Approach .............................................. 95
    4.1.3 Equipment ............................................ 96
    4.1.4 Experimental Setup ................................... 99
    4.1.5 Data Processing ...................................... 100
    4.1.6 Results .............................................. 102
    4.1.7 Discussion .......................................... 104
# List of Tables

3.1 Accuracy of ADV measurements ........................................ 56
3.2 Porous media resistance coefficients associated with the different thickness of porous media. ...................................... 64
3.3 Comparing drag- and lift force on the net panel and velocity reduction 2.5m behind the center of the net panel. .................... 65
3.4 Different cases tested ...................................................... 65
3.5 Different cases tested ...................................................... 65
3.6 Porous coefficients used ................................................... 66
3.7 Comparing results from the different cases .......................... 66
3.8 Different cases tested using a 50mm thick porous media .......... 69
3.9 Different turbulence cases tested using a 50mm thick porous media ................................................................. 72
3.10 Values of $r_n$ found from running CFD simulations at different angles of attack .......................................................... 78
3.11 Porous coefficients from the different error functions .......... 79
3.12 Porous coefficients for the different data sets from Rudi et al. (1988) .......................................................... 84
3.13 Angles of attack and speeds used in the simulation runs that are compared to measured data .......................................... 86

4.1 Drag force measurements ................................................. 103
4.2 Normalized velocity measurements for the double point tow configuration ................................................................. 104
4.3 Porous media resistance coefficients for the higher solidity net ... 108
4.4 Meshes tested and obtained drag force on the cage ............. 109
4.5 Model settings tested and obtained drag force on the cage ..... 114
4.6 Turbulence quantities tested, and obtained drag force on the cage . 118
4.7 Porous resistance coefficients tested, and obtained drag force on the cage . 118
4.8 Porous resistance coefficients tested, and obtained drag force on the porous media . 123
4.9 Turbulence quantities tested, and obtained drag force . 125
4.10 Porous coefficients for the biofouled net . 129
4.11 Turbulence in full scale . 130
4.12 Numerical results from simulations . 134
# List of Figures

1-1 A typical tow tank setup ........................................ 4
1-2 The net is modeled as a thin volume with added resistance .... 8

2-1 The different Re regimes for porous flow ........................ 22
2-2 Two coordinate systems that differ by a rotation ............... 24
2-3 The distance through the porous media changes with angle of attack 25
2-4 Control volumes used to illustrate discretization of a scalar transport equation .................................................. 30
2-5 Tetrahedral cell ..................................................... 42
2-6 Control volume used to calculate forces on a net panel in current ................................................................. 47

3-1 Overview of the setup for measuring forces on the net panel ... 53
3-2 Pictures of the setup for measuring forces on the net panel ... 54
3-3 Loadcell arrangement ................................................ 55
3-4 Instrumentation for measuring carriage speed and current velocity . 55
3-5 The net used for the tests ............................................ 57
3-6 Time series of drag and lift force and measured current ....... 59
3-7 Drag and lift coefficients as a function of angle of attack .......... 60
3-8 Current reduction behind the net panel. ........................... 61
3-9 Velocity reduction across the net panel wake for different meshes ... 70
3-10 Velocity reduction across the net panel wake for different inlet turbulence ............................................................. 73
3-11 The variation of $r_n$ with angle of attack .......................... 78
3-12 Data from the fit using LSNE .................................... 80
3-13 Data from the fit using LAE ....................................... 81
3-14 Data from the fit using LANE  ............................................ 82
3-15 The variation of r with angle of attack  .............................. 83
3-16 Data from the fit of the data from Rudi et al. (1988) using LANE  . 85
3-17 Comparing CFD results to data ............................................ 87
3-18 Sensitivity of analytical method to offset in porous resistance coefficients ......................................................... 89
3-19 Sensitivity of CFD method to offset in porous resistance coefficients  90

4-1 Overview of a gravity cage .................................................... 93
4-2 Overview of the cage ........................................................... 94
4-3 Cage and tow setup ............................................................. 95
4-4 Overview of the tow tank ..................................................... 96
4-5 The current meter calibration setup ....................................... 98
4-6 The tow setup used ............................................................. 99
4-7 The cage deployed in the tank ................................................ 101
4-8 Porous resistance coefficients as a function of solidity ............. 107
4-9 The geometry of the cage model ........................................... 109
4-10 Mesh used for CFD of small cage ......................................... 110
4-11 Velocity distribution around and inside the cage ................. 111
4-12 Pressure distribution at the surface level ............................ 112
4-13 Velocity reduction of different meshes ................................. 113
4-14 Velocity reduction of different model settings ..................... 115
4-15 Velocity reduction using different turbulence quantities ....... 117
4-16 Velocity reduction using different porous resistance coefficients 119
4-17 Comparing nets of different thickness parallel to the flow ...... 122
4-18 Velocity in scaled up cage .................................................. 124
4-19 Overview of the full size cage and domain ............................ 128
4-20 The mesh for the full size cage and domain .................................. 130
4-21 Velocity in full size cage ................................................................. 132
ABSTRACT

FLOW THROUGH AND AROUND FISH FARMING NETS

by

Øystein Patursson
University of New Hampshire, May, 2008

Computational fluid dynamics (CFD) modeling, tow tank and field measurements were used to investigate current flow through and around net panels and cages. For the numerical computations a porous media model was used to represent the net allowing efficient computation of both exterior and interior flow fields. The model was calibrated using tow tank measurements on a net panel at different velocities and angles of attack. The CFD method was able to reproduce the drag- and lift coefficients of the net panel and the velocity reduction behind the net panel with satisfactory accuracy.

The approach was validated for a small size gravity cage by comparing CFD predictions with tow tank measurements of drag force on the cage and velocity reduction inside the cage and in the wake region. The modeled drag force was higher than the measured drag force. The modeled current compared well with the measured current inside the cage, but the reduction was underpredicted in the wake of the cage.

Full scale simulations were performed for a cage with a clean net and a biofouled net and compared with field measurements of a cage fouled with jellyfish. The
measured data compared well with model predictions for the biofouled net. Flushing rates were calculated for both the clean and the biofouled net cases. When the net was changed from clean to biofouled, flushing time increased by up to 44% and drag force increased by up to 80%.
CHAPTER 1

INTRODUCTION

1.1 Cage Flow Problem

The fluid mechanics of flow inside and in the vicinity of fin fish net pens was investigated using computational fluid dynamics (CFD), tow tank testing of net panels and cage models, as well as field observations. Experimental work was used in the development and validation of the CFD model. Velocity reduction within the cage (especially) and in the cage wake was of particular interest. This problem has become critical in the progress of fish farm development for both exposed and sheltered sites since velocity reduction affects drag loads, deformed shape and flushing of the cage.

1.1.1 Background

Fish farming has been used as a food source for a very long time. It is, for example, known to have been practiced in ancient China. In many parts of the world, fish farming originated later and is an addition to the capture fisheries that seem to be exploited to their maximum. According to FAO (2004) the capture fisheries throughout the world are stagnating, and the increased need for fish production is supplied from aquaculture.

Most of the fish farming in the North Atlantic is based on farming of salmonides. This is mainly done using large gravity cages, and the trend has been that the cages
are growing in size, and the biomass on each site is growing as well. In exposed sites, the cages are generally circular cages that range from 80 to 130 m in circumference with a cylindrical net that usually has up to 15 m deep sides and a bottom net that might sag down another 10 m. Lately, 150 m circumference cages have been introduced in Norway and other places as well. Cages used in the tuna industry are even larger, up to 90 m diameter (280 m circumference). The structural part of the cages is generally only a flotation rim that is comprised of one to three High Density PolyEthylene (HDPE) flotation pipes. The rim supports a number of stanchions and a handrail. The stanchions are made from either steel or HDPE and the handrail is generally made of a HDPE pipe of smaller diameter than the flotation pipes. Different solutions for weighting the bottom of the cage are applied; weights integrated in the net or small weights attached to the net are most common on sites experiencing relatively slow currents, while a bottom weight ring is gaining popularity on sites with stronger currents. Sometimes other materials are used for the flotation rim such as rubber hose. The weight ring is of approximately the same size as the top rim and is ballasted such that it’s weight gives the net it’s shape. This means that the structure is mostly made up of net, and any modeling work done on large gravity cages has to emphasize modeling the net correctly.

When a net section is subject to current, the interaction between the net and the fluid causes a drag force (parallel to the current) and, depending on angle of attack and symmetry, possibly also a lift force (perpendicular to the current). The flow field also changes due to this interaction. The drag force on the net is a key parameter in the design of cages and moorings. It is dependent on the net geometry and fluid velocity, and therefore it is important not only to know the forces on the net, but also to know the effect of the cage/net on the flow field, such that these changes in the fluid flow can be included when calculating the forces.

Commercial salmon farms have generally large biomasses, usually from 1000
metric tons to over 5000 metric tons per site. The trend is to have larger and larger sites with more biomass, to have more economical farming, which means that either the fish concentration in the cages is increasing or the number of cages or the size of the cages is increasing. This puts a lot of pressure on the ecological system on and around the site. Good knowledge of the flow field is important when analyzing the ecological environment inside and around the cages. Knowing the flow field inside the cage, and maybe between cages in systems of cages, will give an opportunity to model oxygen concentration and effluent transport inside the cage and through systems of cages. The flushing time, which describes the exchange of water in the cage as the ratio between the volume of the cage and the rate of water flow into the cage, gives a good first basis for comparing two situations with regards to transport of oxygen and effluents.

1.2 Previous Technical Work

Some previous work has been done regarding flow through screens or nets. This work includes both experimental work and theoretical work.

The more basic work regarding fish farming nets has been tow tank testing of flat net panels. Usually such research is carried out by making tow tank tests on net panels stretched on a frame made out of pipe. A typical tow tank setup is shown in Fig 1-1. The main characteristics of the net that are investigated are drag and lift coefficients \((C_d \text{ and } C_l)\) and current reduction \((U_r)\), which are defined by

\[
C_d = \frac{D}{\frac{1}{2} \rho A u_0^2}, \tag{1.1}
\]

\[
C_l = \frac{L}{\frac{1}{2} \rho A u_0^2}. \tag{1.2}
\]
Figure 1-1: A typical tow tank setup. The frame with the net stretched on it can be rotated around the structural member and the drag and lift forces are measured in the structural member. The angle of attack is denoted by $\alpha$.

and

$$ U_r = 1 - \frac{u}{u_0} \quad (1.3) $$

where $D$ is the drag force, $L$ is the lift force, $\rho$ is the density of the fluid, $A$ is the area of the net panel, $u_0$ is the incident velocity and $u$ is the local current velocity. Løland (1991) and Le Bris and Marichal (1998) have published such results.

Theoretical work is often based on experiments. The work of Taylor (1944), Schlichting (1968) and Koo and James (1973) has been used by Løland (1991); he combined theoretical work with the experimental work by Rudi et al. (1988) to derive formulas for drag and lift coefficients ($C_d$ and $C_l$) and wake current reduction ($\left(U_r\right)_{wake}$) of net panels. These formulas are

$$ C_d = 0.04 + (-0.04 + 0.33S + 6.54S^2 - 4.88S^3) \cos \alpha' \quad (1.4) $$
\[ C_l = (-0.055 + 2.3S^2 - 1.765S^3) \sin(2\alpha') \]  
\[ (U_r)_{\text{wake}} = 0.46C_d \]

where \( S \) is the solidity ratio of the net (outline area / thread projected area), \( \alpha' = \frac{\pi}{2} - \alpha \), and \( \alpha \) is the angle of attack of the net panel. Løland (1991) states that the wake behind the net panel is approximately uniform through the width of the net panel and does not change significantly with distance behind the net.

Fredheim (2005) builds further on the work by Løland (1991) by calculating the flow distribution around a three dimensional structure of net as a superposition of effects due to individual threads and knots. The disturbance on the flow field by the cylinders and spheres describing the twine and knots of the net is in two parts. The wake is described by the "far wake mean velocity deficit" model first presented by Schlichting (1968), with modifications such that it can better be applied in the near field of the cylinders. The disturbance on the flow field outside the wake region by the presence of the cylinders and spheres and the wake generated by them is described by a distribution of sources. The strength of the sources is related to the drag force on the elements by Lagally's theorem (Milne-Thomson, 1968).

Tank tests of cages have also been performed. Rudi et al. (1988) have measured drag force and current reduction on square cages, both single cages and systems of up to 6 cages. The size of the cages where current reduction was measured was 1.5m by 1.5m by 1m, and the cages were placed in 2 rows of 3 cages aligned with the flow. The measured data in the cage centers and in the wake are compared to the formulas by Løland (1991) for current reduction (Equation 1.6), and good agreement was found.

Vincent and Marichal (1996) have used axisymmetric computational fluid dy-
namics to model the flow through a trawl codend and compared predictions to tank measurements of flow through a horizontal cylinder and a horizontal cone made of net. They model the net as a permeable surface with normal flow \( u_n \) given by

\[
    u_n = \frac{\Delta p}{\rho u_0}
\]  

(1.7)

and tangential flow \( u_t \) as

\[
    u_t = \frac{\beta}{\rho u_0} \tau_{eff}^+ + \tau_{eff}^-
\]  

(1.8)

with

\[
    \tau_{eff} = (\mu + \mu_t) \frac{\partial u_t}{\partial n}.
\]  

(1.9)

In these expressions \( \alpha \) and \( \beta \) are resistance coefficients, \( u_0 \) is the undisturbed velocity, \( \mu \) and \( \mu_t \) are viscous and turbulent viscosities and \( \tau_{eff} \) is the effective stress on the internal face (+) and the external face (−) of the net. The derivative with respect to \( n \) is a spatial derivative where the coordinate \( n \) is normal to the net.

Using CFD seems to be a promising approach to model flow through and around nets. Patursson et al. (2006) have investigated another method using a porous media model that is already existing in CFD-packages to model the net. The method is described in Chapter 2, and previous work using net panels indicated that the approach seems promising.

Full size measurements have also been made of net structures. Patursson and Simonsen (2008) have made measurements of net deformation and current reduction inside and in the wake of a full size circular gravity cage (30m diameter and 12m deep) deployed on a site with relatively strong tidal currents. The net was free of regular biofouling (algae or shellfish growth), but a large number of jellyfish on the net increased the solidity such that the cage deformation and velocity reduction was rather large compared to the numbers expected for a clean net.
Johansson et al. (2007) have performed current measurements at four different farms in Norway. Currents were measured upcurrent and downcurrent of the cage systems and at reference stations to the side of the cage systems. Two cages were between the upcurrent and the downcurrent positions. In three of four cases the upcurrent measurement was lower than the reference station, and in all cases a significant current reduction was observed between the downcurrent and the upcurrent position. The observed reduction was between 33% and 64%.

Vincent and Marichal (1996) measured the current velocity inside a trawl and wanted to compare the velocity inside the trawl to the velocity away from the trawl to investigate the effect of the trawl on the flow field. This proved to be difficult, and unknown effects made the measurements hard to use for accurate comparing against simulations.

Field measurements are usually associated with a lot of uncertainties, and therefore tank tests are usually preferred when possible. This is especially important when measuring velocity reduction, since the measurement depends on two independent current measurements generally of similar magnitude, and small errors in the measurements can have a large effect on the measured current reduction.

1.3 Goals

The goals of the dissertation research are:

- Use CFD to model flow through nets.
- Calibrate and validate the model using tow tank measurements on net panels.
- Model flow through gravity cages using CFD and validate against tow tank measurements and field measurements.
- Calculate flushing rates for cages using modeled currents inside cages.
Figure 1-2: The net is modeled as a thin homogeneous volume with added resistance.

- Apply modeling approach to biofouled cages making appropriate assumptions.

1.4 Approach

Flow through nets were modeled using three-dimensional (3D) CFD, where the net was described as a thin volume with added resistance. The added resistance was prescribed using a porous media formulation. The approach is presented graphically in Figure 1-2.

Tow tank tests of a net panel at different angles of attack were performed at the University of New Hampshire (UNH) where drag and lift force on the net panel and current reduction behind the net panel were measured as functions of speed and angle of attack. These measurements were used, in addition to theoretical considerations and CFD results, to develop a method to find optimal model constants for a net panel.

Tow tank tests at the U. S. Naval Academy (USNA) were performed on a small gravity cage to generate data to validate the model. The model was applied to
the gravity cage using optimized model constants from net panel tests. Model evaluation involved comparing prediction and measurements of velocity distribution and drag. Sensitivity of predictions due to mesh, turbulence assumptions and resistance coefficients was explored.

The model was also applied and compared to available measurements of current reduction inside and behind a full scale cage by Patursson and Simonsen (2008). The measurements by Swift et al. (2006) were used to estimate porous media resistance coefficients for the jellyfish-fouled net, and models were run for the full scale cage with and without fouling. Velocity reduction, drag force on the cage and flushing rates were compared.
CHAPTER 2
THEORETICAL CONSIDERATIONS

Computational fluid dynamics (CFD) are generally used by engineers to solve fluid dynamics problems that involve solving some form of the Navier-Stokes equations. When modeling the flow through and around fish farming cages, the flow through the cages depends on how much of the flow is diverted around the cages, and the most general way of approaching the problem is to model the flow for a large domain enclosing the cages. Due to the 3D nature of the geometry, a full 3D model is required. The large dimensions (cages ~ 30m and net panel ~ 1m) cause large Reynolds numbers (Re) at slow speeds, which means that the flow must be assumed turbulent and a turbulence model needs to be applied. The Re can be described as the ratio between inertial and viscous forces in a flow

\[ Re = \frac{\rho u D}{\mu} \]  

(2.1)

where \( \rho \) and \( \mu \) are the density and the viscosity of the fluid, \( u \) is the fluid velocity and \( D \) is the characteristic dimension of the structure. In addition some way of describing the resistance of flow through the net is necessary. These requirements are met in the CFD software package FLUENT 6.3 from Fluent Inc. when modeling the net as a thin volume with added resistance described by the porous media model included in the package and by applying one of the turbulence models.

CFD packages generally do not include the Coriolis force. Although FLUENT 6.3 had the possibility to use rotating reference frames it was chosen to not include the Coriolis force in the present work. The importance of the Coriolis force can be
described by the Rossby number

\[ R_0 = \frac{U}{L f}, \]  

(2.2)

where \( U \) is current speed, \( L \) is length scale and \( f = 2\Omega \sin \phi \) where \( \Omega \) is the angular velocity of the earth and \( \phi \) is latitude. If the Rossby number is large, the Coriolis force is small compared to other forces.

If a full system of maybe 10 cages was to be modeled, the domain would be rather large, \( L > 500m \). Using this domain size and a slow, but realistic, speed of \( 5\text{cm s}^{-1} \) at \( 62^\circ \) latitude, \( R_0 < 0.78 \). This might imply that the Coriolis force would be of importance, but the main interest lies in what happens inside and between the cages. Using those length scales, \( L < 50m \), \( R_0 \) would be fairly large and the Coriolis force would have little importance and could be neglected.

2.1 Hydrodynamic Equations Solved

The equations solved were the momentum equation, also known as the Navier-Stokes equation, and the continuity equation. Incompressible flow was assumed so that the equations were (in tensor notation):

\[ \frac{Du_i}{Dt} = \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + g_i + \frac{1}{\rho} \frac{\partial \tau_{ij}}{\partial x_j} + S_i \]  

(2.3)

and

\[ \frac{\partial u_i}{\partial x_i} = 0 \]  

(2.4)

where \( \tau_{ij} = 2\mu S_{ij}, S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \), \( g_i = (0, 0, -g) \), \( g \) is the acceleration due to gravity and \( S_i \) is a term to take external body forces into account. In Equations (2.3) and (2.4) \( u_i \) is the three-dimensional velocity vector, \( \rho \) is the mass density, \( \tau_{ij} \) is the stress tensor, \( \mu \) is the viscosity and \( S_{ij} \) is the mean strain rate tensor.
The shown set of equations is useful for solving laminar flows, but for turbulent flows the direct solving of all the vortices in a turbulent flow is very expensive. Therefore a model for turbulence needed to be added. There are several types of turbulence models available. The most common are the Reynolds averaged Navier Stokes (RANS) models and large eddy simulation (LES) models. The LES models resolve eddies due to turbulence down to a certain size and the effect of the smaller eddies is modeled. The RANS models only resolve the ensemble averaged flow and model the effect of the turbulent eddies. Therefore the LES models require much more computational power than the RANS models. In the present study the focus was on describing the resistance of the net and to a lesser extent to evaluate turbulence models and the turbulence model used needed to be as simple and robust as possible. These criteria were thought to be met in the two-equation RANS models. For the present study three of the two-equation RANS models were tested. Further work with turbulence models was thought to be outside the scope of this study, but a test using LES would be interesting and could be part of the later work in the area.

2.1.1 Modeling Turbulence through Reynolds Averaging

Solving the Navier-Stokes equations with Reynolds averaging (RANS) is a commonly used approach and is explained in many textbooks (e.g. Pope (2000)). If Reynolds averaging is employed, the instantaneous velocity can be taken as the combination of the mean \( \bar{u}_i \) and fluctuating components \( u'_i \), then \( u_i = \bar{u}_i + u'_i \). If the same holds true for the pressure \( P \), the averaging of Equation (2.3) and dropping the overbars over the mean pressure and mean velocity results in the Reynolds averaged Navier-Stokes equation,

\[
\frac{Du_i}{Dt} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + g_i - \frac{\partial \bar{u}_i u'_j}{\partial x_j} + \nu \frac{\partial}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \frac{1}{\rho} S_i, \quad (2.5)
\]
where $v = \frac{\mu}{\rho}$. The continuity equation becomes

$$\frac{\partial u_i}{\partial x_i} = 0. \quad (2.6)$$

Equation (2.5) represents a balance between the change in momentum and the forces acting on the fluid - pressure, gravity, Reynolds stress, viscous diffusion and body forces, respectively.

The RANS models are divided up into classes depending on the number of variables and hence equations describing the Reynolds stress. E.g. one equation models, two equation models and Reynolds stress models where all the elements of the Reynolds stress tensor are modeled. There are to the knowledge of the author no exact guidelines stating what models perform best for different types of flow, but generally higher levels of complexity in the model will provide more accurate results. For the present work it was decided to use two equation RANS turbulence models due to their reasonable accuracy and completeness and low computational cost. The two equation models are the simplest RANS models where the turbulent velocity and length scales are independently determined. The model proposed by Launder and Spalding (1972), usually termed the standard $k - \epsilon$ model (SKE), is generally accepted to give accurate results for simple flows. However it is well known that for certain flows the standard $k-\epsilon$ model produces inaccurate results. One of the best known is the inability to predict the spreading rate of round jets (Shih et al., 1995; Pope, 2000). Several refinements have been made to the SKE (e.g. Yakhot and Orszag (1986) and Shih et al. (1995)). The main difference is usually within the $\epsilon$ transport equation and the eddy viscosity formulation. The model proposed by Shih et al. (1995) (when implemented in FLUENT 6.3 it is termed the realizable $k - \epsilon$ model (RKE)) is shown to be superior to the SKE for most flows (Shih et al., 1995) and more accurately predicts the spreading rate of planar and
round jets. For the present work the SKE and the RKE models were used for most simulations, but the standard $k - \omega$ turbulence model (SKO) proposed by Wilcox (1998) was also tested.

Generally the two equation RANS models use a linear eddy viscosity model, where the closure is achieved using a Boussinesq-type approximation between the Reynolds stress and the strain rate so that

$$\overline{u'_i u'_j} = \frac{2}{3} k \delta_{ij} - \nu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right). \quad (2.7)$$

In Equation 2.7 $\nu_t = \frac{\mu_t}{\rho}$ where $\mu_t$ is the eddy viscosity and $k$ is the turbulent kinetic energy, which describes the magnitude of the turbulent velocity fluctuations. Incorporating (2.7) into (2.5), yields,

$$\frac{D u_i}{D t} = \frac{1}{\rho} \frac{\partial p}{\partial x_i} + g_i + \frac{\partial}{\partial x_j} \nu_{eff} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \frac{1}{\rho} S_i \quad (2.8)$$

where $p = P + \frac{2}{3} \rho k$ and $\nu_{eff} = \nu + \nu_t$.

$\nu_t$ is a function of $k$ and $\epsilon$ or $\omega$, which are the turbulent dissipation rate or the specific dissipation rate, respectively. The turbulent dissipation rates describe the rate at which $k$ dissipates. Equations describing those variables are needed for the closure of the equation system. These equations are usually generated by forming scalar transport equations for the wanted variables. The scalar transport equation used is the following

$$\frac{\partial \rho \phi}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i \phi) = \frac{\partial}{\partial x_i} \left( \Gamma_\phi \frac{\partial \phi}{\partial x_i} \right) + S_\phi, \quad (2.9)$$

where $\phi$ is the scalar, $\Gamma_\phi$ is the diffusion coefficient for the scalar $\phi$ and $S_\phi$ is the source term for the scalar. One such equation is then set up for each of the variables $k$ and $\epsilon$ using appropriate expressions for $\Gamma_\phi$ and $S_\phi$. 

14
2.1.2 The Standard $k - \epsilon$ Model

The SKE has become the workhorse for practical engineering modeling of turbulent flows due to its robustness, economy and reasonable accuracy for a wide range of flows. The model is semi empirical, where the transport equation for $k$ is derived from the exact equation while the equation for $\epsilon$ has less resemblance to its exact mathematical formulation.

In the derivation of the $k$-$\epsilon$ model, it was assumed that the flow was fully turbulent, and the effects of molecular viscosity were negligible. The standard $k$-$\epsilon$ model is therefore valid only for fully turbulent flows.

The equations for $k$ and $\epsilon$ used in the standard $k$-$\epsilon$ model are

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \tag{2.10}
\]

\[
\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_i} (\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_3 \epsilon G_b) - C_2 \epsilon \rho \frac{\epsilon^2}{k} + S_\epsilon. \tag{2.11}
\]

In the $k$-$\epsilon$ models the turbulent (or eddy) viscosity, $\mu_t$, is computed by combining $k$ and $\epsilon$ as follows:

\[
\mu_t = \rho C_\mu \frac{k^2}{\epsilon}. \tag{2.12}
\]

The model constants in the standard $k$-$\epsilon$ model have the following default values: $C_{1\epsilon} = 1.44$, $C_2 = 1.92$, $C_\mu = 0.09$, $\sigma_k = 1.0$ and $\sigma_\epsilon = 1.3$.

The term $G_k$, representing the production of turbulence kinetic energy, is modeled identically for the available $k$-$\epsilon$ models. From the exact equation for the transport of $k$, this term may be defined as
Applying the Boussinesq hypothesis gives,

\[ G_k = -\rho u_i u_j \frac{\partial u_j}{\partial x_i}. \]  

(2.13)

where \( S \) is the modulus of the mean rate-of-strain tensor, defined as

\[ S \equiv \sqrt{2S_{ij}S_{ij}}. \]  

(2.15)

The terms \( G_b \) and \( Y_M \) due to buoyancy and compressibility were not necessary when modeling incompressible flow with no temperature fluctuations, hence \( G_b = 0 \) and \( Y_M = 0 \). \( S_k \) and \( S_c \) are source terms that can be defined by the user.

### 2.1.3 The Realizable \( k - \epsilon \) Model

The realizable \( k - \epsilon \) model is a relatively recent development. It was proposed by Shih et al. (1995) and was intended to address deficiencies of the traditional \( k - \epsilon \) models by applying a new eddy viscosity formula involving a variable \( C_\mu \) originally proposed by Reynolds (1987) and a new transport equation for the dissipation rate, \( \epsilon \), which has been derived based on the dynamic equation of the mean-square vorticity fluctuation.

The term "realizable" means that the model satisfies the mathematical constraints on the Reynolds stresses: that the normal Reynolds stress is a positive quantity and that the Schwarz inequality \((u_\alpha u_\beta)^2 \leq u_\alpha^2 u_\beta^2\), no summation over \( \alpha \) and \( \beta \) is valid. The standard \( k - \epsilon \) model is not realizable. An immediate benefit of the realizable \( k - \epsilon \) model is that it more accurately predicts the spreading rate of both planar and round jets. It is also likely to provide superior performance for
other complex flows. The realizable k-ε model is still relatively new. However, the studies so far indicate that the RKE has much better performance than the SKE (Shih et al., 1995; Fluent, 2006).

The transport equation for $k$ used in the realizable $k-\epsilon$ model is the same as used in the standard $k-\epsilon$ model (Equation 2.10), and the equation for $\epsilon$ is:

$$\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_i} (\rho \epsilon u_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_1 S_\epsilon - \rho C_2 \frac{\epsilon^2}{k} + \frac{\rho C_3}{k} \frac{\epsilon}{\epsilon} (\nabla \cdot \vec{u}) + S_{\epsilon}$$

(2.16)

where $C_1 = \max \left[ 0.43, \frac{\eta}{\eta + 5} \right]$, $\eta = S_{\epsilon}^{1/2}$, $S = \sqrt{2S_{ij}S_{ij}}$.

As in other $k-\epsilon$ models, the eddy viscosity is calculated from Equation 2.12, but $C_\mu$ is not a constant. It is computed from

$$C_\mu = \frac{1}{A_0 + A_s \frac{kU^*}{\epsilon}}$$

(2.17)

where

$$U^* \equiv \sqrt{S_{ij}S_{ij} + \tilde{\Omega}_{ij}\tilde{\Omega}_{ij}}$$

(2.18)

and

$$\tilde{\Omega}_{ij} = \Omega_{ij} - 2\epsilon_{ijk}\omega_k$$

(2.19)

$$\Omega_{ij} = \tilde{\Omega}_{ij} - \epsilon_{ijk}\omega_k$$

(2.20)

where $\tilde{\Omega}_{ij}$ is the mean rate-of-rotation tensor viewed in a rotating reference frame with the angular velocity $\omega_k$. The model constants $A_0$ and $A_s$ are given by $A_0 = 4.04$, $A_s = \sqrt{6\cos\phi}$ where $\phi = \frac{1}{3} \cos^{-1} (\sqrt{6W})$, $W = \frac{S_{ij}S_{jk}S_{ki}}{S^2}$, $\tilde{S} = \sqrt{S_{ij}S_{ij}}$, $S_{ij} = $
The model constants have been defined for best overall performance. The model constants are $C_1 = 1.44$, $C_2 = 1.9$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.2$.

2.1.4 The Standard $k - \omega$ Model

The standard $k - \omega$ model (SKO) has only been used for one test in the following work and is therefore not described in detail, but the equations used are shown below. It is based on the model proposed by Wilcox (1998) and it is generally not considered to be as robust as the SKE, but it has some advantages especially close to the wall boundary condition (Fluent, 2006). This was not the main focus in the present work, but the model was tested to get an indication of how it performed when predicting the exterior flow behind a cage.

The basic difference between the SKO and the SKE is that in the SKO $\omega$ is modeled to predict the dissipation of turbulence. There are different options for adapting the model. The high Re version was used, such that the damping function available was not employed. A shear flow correction was employed and since the flow was incompressible the compressibility correction was not effective.

The eddy viscosity is calculated

$$\mu_t = \frac{\rho k}{\omega}. \quad (2.21)$$

The transport equation for $k$ is

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho ku_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k, \quad (2.22)$$
and the transport equation for $\omega$ is

$$\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + S_\omega. \quad (2.23)$$

The production terms are

$$G_k = \mu t S^2 \quad (2.24)$$

and

$$G_\omega = \frac{\omega}{k} G_k \quad (2.25)$$

where

$$\alpha = \frac{\alpha_\infty (\alpha_0 + \text{Re}_t / \text{Re}_\omega)}{\alpha^* \left( 1 + \text{Re}_t / \text{Re}_\omega \right)} \quad (2.26)$$

and

$$\text{Re}_t = \frac{\rho k}{\mu \omega}. \quad (2.27)$$

The dissipation terms are for $k$

$$Y_k = \rho \beta^* f_{\beta^*} k \omega \quad (2.28)$$

where

$$f_{\beta^*} = \begin{cases} 
1 & \chi_k \leq 0 \\
\frac{1 + 680 \chi_k^2}{1 + 400 \chi_k^2} & \chi_k > 0 
\end{cases} \quad (2.29)$$

and

$$\chi_k \equiv \frac{1}{\omega^3} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}. \quad (2.30)$$

and for $\omega$
\[ Y_\omega = \rho \beta f_\beta \omega^2 \]  

(2.31)

where

\[ f_\beta = \frac{1 + 70 \chi_\omega}{1 + 80 \chi_\omega}. \]  

(2.32)

\[ \chi_\omega = \frac{\Omega_{ij} \Omega_{jkl} S_{ki}}{(\beta \omega)^3} \]  

(2.33)

and

\[ \Omega_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right). \]  

(2.34)

The model constants are \( \alpha_\infty = 0.52, \alpha^* = 1, \alpha_0 = 1/9, R_\omega = 2.95, \sigma_k = 2, \sigma_\omega = 2, \beta^* = 0.09 \) and \( \beta = 0.072 \).

### 2.1.5 Flow through Porous Media

The net was described as a narrow volume region with added resistance modeled as flow through porous media. Flow through porous media is usually described as a pressure drop that is a function of current velocity following the equation proposed by D'arcy (1856) and later revised by Forchheimer (1901). In vector form it is

\[ \mathbf{I} = \mathbf{A} \mathbf{u} + B |\mathbf{u}| \mathbf{u} \]  

(2.35)

where \( \mathbf{I} \) is a hydraulic gradient (gradient of hydraulic head, \( \mathbf{I} = \frac{1}{\rho g} \nabla p \)), \( \mathbf{u} \) is the velocity and \( A \) and \( B \) are constants. The first term on the right hand side of the equation is associated with low Re flow or Darcian flow, and was introduced by D'arcy (1856), while the second term introduced by Forchheimer (1901) is asso-
associated with higher Re flow. The parameter $A$ is often written as $A = \frac{\nu}{gk}$, where $\nu$ is the kinematic viscosity of the fluid and $k$ is the permeability of the porous media. Burcharth and Andersen (1995) described the use of Equation 2.35, when used for flow through porous media, by means of different flow regimes obtained by experiments using stacked transparent spheres as the porous media by Dybbs and Edwards (1984).

**The Darcy Flow Regime** The flow is characterized by creeping flow. $Re < 1$. Only the first term in Equation 2.35 is significant.

**The Forchheimer Flow Regime** Boundary layers are present, and an "inertial core" flow is present between the boundary layers that introduces a nonlinear relationship between pressure drop and fluid velocity. Hence both terms in Equation 2.35 are significant. This is a transition between the Darcy flow regime and the fully turbulent flow regime.

**The Fully Turbulent Flow Regime** A highly unsteady and chaotic flow regime that qualitatively resembles turbulent flow. $Re > 300$. In this flow regime the second term in Equation 2.35 is dominating, but usually both terms are used to describe the flow.

It is important to note that the three flow regimes are usually described using different sets of porous media resistance coefficients as illustrated in Fig. 2-1. The Re-ranges mentioned by Burcharth and Andersen (1995) are approximate and no clear definition is given what the characteristic dimension is, but it can be assumed that probably grain size or maybe pore size is used. For the net panel studies, a comparable Re would therefore be calculated using either twine diameter or mesh size as the characteristic dimension. Using a slow speed of 10 cm s$^{-1}$, which was the slowest possible speed for the tow tests, and a twine diameter of 2 mm, the Re >
Figure 2-1: The different Re regimes for porous flow. Note that the y-axis variable is $\frac{L}{u}$ and the x-axis variable is Re or fluid velocity. Figure is from Burcharth and Andersen (1995).

200. This means that if the same approach and Re range is applicable to the net panel tests, the flow should be described by the Forchheimer and the fully turbulent flow regimes.

When modeling flow through porous media using CFD a similar expression to Equation 2.35 can be included in the momentum equation (Equation 2.3 or 2.8) using the external body force term ($S_i$). $S_i = 0$ outside the porous media and inside the porous media it is described by the following equation

$$S_i = -D_{ij} \mu u_j - \frac{1}{2} C_{ij} \rho u_{mag} u_j$$  \hspace{1cm} (2.36)$$

where $\mu$ and $\rho$ are viscosity and density of the water and $D_{ij}$ and $C_{ij}$ are prescribed material matrices consisting of the porous media resistance coefficients in local x, y and z-direction. The local axes are the principle axes of the porous media. The
matrices are quadratic and the porous media resistance coefficients are placed in the diagonal of the matrix starting with the x-direction coefficient. For a homogeneous porous media all elements in the diagonal are of equal value. If the local axes of the porous media are not aligned with the global coordinate axes, a vector rotation approach has to be employed. Using the terminology in Fig. 2-2 where the primed coordinate system is the global coordinate system, Equation 2.36 can be expressed as

\[ S'_i = -\mu D'_{ij} u'_j - \frac{1}{2} \rho u_{mag} C'_{ij} u'_j \] (2.37)

where

\[ D'_{ij} = R_{ip} R_{jq} D_{pq} \] (2.38)

and

\[ C'_{ij} = R_{ip} R_{jq} C_{pq} \] (2.39)

where \( R_{ij} \) is the rotation matrix, consisting of the direction cosines of the primed axes \((X'_i)\) with respect to the unprimed axes \((X_i)\).

Modeling flow through porous media using Equations 2.8 and 2.36, the pores in the porous media have no influence on the turbulence, and the turbulence inside the porous media is calculated just as in a free flow without obstructions. If the porous media is surrounded by open flow, the only effect on turbulence is the added shear in the flow due to the resistance to the free flow by the porous media. In porous media with low permeability this is probably a significant error since all large vortices would be killed in the porous media. Flow through a net has a smaller effect on the turbulence, but vortices larger than the mesh size will likely be damped by the net, while the net twines will create small vortices at high enough \( \text{Re} \).
As can be seen in Patursson et al. (2006) when modeling the net using the above approach, it has to be considered an anisotropic porous media with different porous media resistance coefficients tangential to the net than normal to the net. Part of the explanation for this lies in the fact that the distance a fluid particle has to travel through the volume describing the net panel depends on angle of attack as demonstrated in Fig. 2-3. Another factor that makes the assessment of the tangential pressure drop coefficients nontrivial is that for flow normal to the net, the geometrical model of the net is of the same size as the real net, while for the flow in the tangential directions the model of the net might be of a much larger dimension than the thickness of the net. The porous media resistance coefficients are defined as:

**Normal Porous Media Resistance Coefficients** \( (D_n, \text{ and } C_n) \) Porous media resistance coefficients normal to the net panel, say it was the x-direction.
Figure 2-3: Sketch showing how the distance $\Delta x$ through the volume of porous media describing the net panel changes with angle of attack $\theta$.

in the local coordinate system.

**Tangential Porous Media Resistance Coefficient ($D_t$ and $C_t$)** Porous media resistance coefficients in the tangential directions of the net. If the normal porous media resistance coefficients were in the local $x$-direction, then the tangential porous media resistance coefficients were in the local $y$- and $z$-directions. They were assumed equal in both directions.

### 2.2 Boundary Conditions

The boundary conditions used for the present study are explained below. The velocity inlet boundary condition is best applied to incompressible flows when the flow velocity at the inlet is known. This was the case for all simulations in the present work and this was the only inlet boundary condition used. The outflow boundary conditions can be described either by the outflow or the pressure outlet. For the steady state cases modeled in the present work, the difference was small and either one could be used. It was chosen to use the pressure outlet for all simulations. Solid surfaces and the water surface were modeled using the wall
boundary condition.

2.2.1 Velocity Inlet Boundary Condition

The mass flow rate entering a fluid cell adjacent to the velocity inlet boundary is computed as

\[ m = \int \rho \bar{u} \cdot d\bar{A}. \]  \hspace{1cm} (2.40)

For turbulent calculations, the inlet turbulence parameters need to be supplied. Estimating the quantities can be done on the basis of the following relationships (Fluent, 2006).

The turbulent kinetic energy, \( k \), can be found from:

\[ k = \frac{3}{2} u'_{\text{rms}}^2 \] \hspace{1cm} (2.41)

where \( u'_{\text{rms}} \) is the root mean square of the turbulent velocity fluctuations, or

\[ k = \frac{3}{2} (\langle u \rangle l)^2 \] \hspace{1cm} (2.42)

where \( l = \frac{u'_{\text{rms}}}{\langle u \rangle} \) is the turbulent intensity. Turbulence intensity of 1% is considered low, while turbulence intensities > 10% are considered high (Fluent, 2006).

The turbulence length scale, \( l \), is associated with the size of the large eddies that contain the energy in turbulent flows. E.g. an approximate relationship between \( l \) and the physical size of a duct, \( L \), in fully developed duct flow is:

\[ l = 0.07L. \] \hspace{1cm} (2.43)

The turbulent dissipation rate, \( \epsilon \) can be determined from \( l \)
\[ \epsilon = C_{\mu}^3 \frac{k^{3/2}}{l} \]  

(2.44)

where \( C_{\mu} \) is an empirical constant in the k-\( \epsilon \) models (Sections 2.1.2 and 2.1.3). For the standard k-\( \epsilon \) model \( C_{\mu} \approx 0.09 \). The value of \( \epsilon \) can also be obtained from the turbulent viscosity ratio \( \frac{\mu_t}{\mu} \)

\[ \epsilon = \rho C_{\mu} \frac{k^2}{\mu} \left( \frac{\mu_t}{\mu} \right)^{-1} \]  

(2.45)

Finally the value of \( \epsilon \) can also be defined from a known or wanted decay of \( k \), \( \Delta k \), over a certain streamwise length of the flowdomain, \( L_{\infty} \),

\[ \epsilon \approx \frac{\Delta k U_{\infty}}{L_{\infty}} \]  

(2.46)

where \( U_{\infty} \) is the free-stream velocity.

\( \omega \) can also be found from similar formulas

\[ \omega = \frac{k^{1/2}}{C_{\mu}^{1/4} l} \]  

(2.47)

\[ \omega = \frac{k}{\rho} \left( \frac{\mu_t}{\mu} \right)^{-1} \]  

(2.48)

### 2.2.2 Pressure Outlet Boundary Condition

At pressure outlets, FLUENT 6.3 uses the specified boundary condition pressure as the static pressure of the fluid at the outlet plane and extrapolates all other conditions from the interior of the domain. For the present work the density was constant, no gravitational force was included and the pressure at the inlet and exit were both, \( p_i = p_o = 0 \).
2.2.3 Wall Boundary Condition

Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, the no-slip boundary condition is enforced at walls by default, but a tangential velocity component can be specified in terms of the translational or rotational motion of the wall boundary, or model a "slip" wall by specifying shear. In the present work tank walls were modeled as moving walls with the same speed as the incoming fluid specified at the velocity inlet boundary condition or in some cases as walls with zero shear force. The water surface was modeled as a wall boundary condition with zero shear force. Other solids were modeled using the regular no slip condition. The shear stress at the walls was calculated as explained in Section 2.2.4.

2.2.4 Near Wall Treatment

For coarse meshes, boundary layers can not always be discretized in sufficient detail. This problem is solved through the use of wall functions where semi-empirical formulas are used to bridge the solution variables in the cell next to the wall to the values at the wall. The wall functions used were the standard wall functions based on the proposal of Launder and Spalding (1974).

Momentum

The wall functions for mean velocity are

\[ U^* = \frac{1}{k} \ln(Ey^*) \]  \hspace{1cm} (2.49)

where

\[ U^* \equiv \frac{U_p(C_\mu^{\frac{4}{3}} k_F^{\frac{1}{3}})}{\frac{2\mu}{\rho}}, \]  \hspace{1cm} (2.50)
\[ y^* = \rho C_{\mu}^{1/4} k_{P}^{3/2} y_{P} \frac{1}{\mu}, \]  

(2.51)

\[ \tau_w = \mu \frac{\partial u}{\partial n} \]  

(2.52)

and

\[ \kappa = \text{von Kármán constant (}= 0.4187) \]

\[ E = \text{empirical constant (}= 9.793) \]

\[ U_{P} = \text{mean velocity of the fluid at point } P \]

\[ k_{P} = \text{turbulence kinetic energy at point } P \]

\[ y_{P} = \text{distance from point } P \text{ to the wall} \]

\[ \mu = \text{dynamic viscosity of the fluid} \]

\[ n = \text{local coordinate normal to the wall}. \]

Generally the logarithmic law for mean velocity is known to be valid for \(30 < y^* < 300\). In the present model it was employed when \(y^* > 11.225\). If the mesh was such that \(y^* < 11.225\) at the wall-adjacent cells, the laminar stress-strain relationship is applied

\[ U^* = y^*. \]  

(2.53)

**Turbulence**

When using the standard wall functions the \(k\) equation is solved in the whole domain including the wall-adjacent cells. The boundary condition for \(k\) imposed at the wall is

\[ \frac{\partial k}{\partial n} = 0 \]  

(2.54)
Figure 2-4: 2 dimensional triangular control volumes used to illustrate discretization of a scalar transport equation. Figure is reproduced from Fluent (2006)

The production of $k$, $G_k$, at the wall-adjacent cells is computed from

$$G_k \approx \tau_w \frac{\partial U}{\partial y} = \tau_w \frac{\tau_w}{\kappa \rho \gamma^4 k^2 \gamma^2 y}$$

and $\epsilon$ is computed from

$$\epsilon_p = \frac{C_p^\gamma k^\gamma}{\kappa y_p}.$$  \hspace{1cm} (2.56)

### 2.3 Finite Volume Discretization

The equations were solved using a finite volume technique where the equations were integrated over a control volume. The control volume integrals were discretized into sums, which yielded discrete equations. The discrete equations were applied to each of the control volumes or cells in the computational domain and solved iteratively.

The two-dimensional, triangular cell shown in Figure 2-4 is an example of such a control volume. A short explanation of the discretization process is given below. More thorough explanations are found in Fluent (2006) and Ferziger and Perić (2002).
2.3.1 Discretization of the Scalar Transport Equation

The approach is first illustrated by discretization of the steady-state version of the conservation equation for transport of a scalar quantity \( \phi \) (Equation 2.9). The control volume integrated steady state version of Equation 2.9 is written as:

\[
\int \rho \phi \vec{v} d\vec{A} = \int \Gamma_{\phi} \nabla \phi d\vec{A} + \int_{V} S_{\phi} dV \tag{2.57}
\]

where

- \( \rho \) = density
- \( \vec{v} \) = velocity vector (= \( u\hat{i} + v\hat{j} \) in 2D)
- \( \vec{A} \) = control volume surface area vector
- \( \Gamma_{\phi} \) = diffusion coefficient for \( \phi \)
- \( \nabla \phi \) = gradient of \( \phi \) (= \( \frac{\partial \phi}{\partial x} \hat{i} + \frac{\partial \phi}{\partial y} \hat{j} \) in 2D)
- \( S_{\phi} \) = source of \( \phi \) per unit volume.

Discretization of Equation 2.57 on a given cell yields

\[
\sum_{f}^{N_{\text{faces}}} \rho_{f} \vec{v}_{f} \phi_{f} \vec{A}_{f} = \sum_{f}^{N_{\text{faces}}} \Gamma_{\phi} (\nabla \phi)_{n} \vec{A}_{f} + S_{\phi} V \tag{2.58}
\]

where

- \( N_{\text{faces}} \) = number of faces enclosing cell
- \( \phi_{f} \) = value of \( \phi \) convected through face \( f \)
- \( \rho_{f} \vec{v}_{f} \vec{A}_{f} \) = mass flux through the face
- \( \vec{A}_{f} \) = area of face \( f \), \( |A| \) (= \( A_{x} \hat{i} + A_{y} \hat{j} \) in 2D)
- \( (\nabla \phi)_{n} \) = magnitude of \( \nabla \phi \) normal to face \( f \)
- \( V \) = cell volume.

The discrete values of the scalar \( \phi \) are stored at the cell centers (c0 and c1 in
Figure 2-4). The face values $\phi_f$ and derivative $\nabla \phi$ are found from the surrounding cell center values as shown in Sections 2.3.2 and 2.3.3 and a linearized form of Equation 2.58 can be written as

$$a_P \phi = \sum_{nb} a_{nb} \phi_{nb} + b$$

(2.59)

where the subscript $nb$ refers to neighbor cells, and $a_P$ and $a_{nb}$ are the linearized coefficients for $\phi$ and $\phi_{nb}$.

The above equation was written for each cell in the grid, and the system of equations was solved using a point implicit (Gauss-Seidel) linear equation solver in conjunction with an algebraic multigrid (AMG) method explained in Section 2.3.8.

### 2.3.2 Discretization Schemes

In FLUENT 6.3 the values of the variables in the equations solved are stored at the cell centers. The values of variables at cell faces are needed when solving the equations. These are found by upwind discretization schemes. The basic idea with upwind schemes is that the face value of a given variable is defined from the cell center value in the cell upstream of the face. Upstream is defined relative to the fluid velocity normal to the face. The first-order upwind and second-order upwind schemes are described below.

**First Order Upwind Discretization Scheme**

When the first order upwind scheme is used, the values of variables at cell face are the values of the same variables at the upwind cell center.
Second Order Upwind Discretization Scheme

When the second order upwind scheme is used, the face value \( \phi_f \) is calculated using a Taylor series approach

\[
\phi_f = \phi + \nabla \phi \cdot \Delta \vec{S}
\]

(2.60)

where \( \phi \) and \( \nabla \phi \) are the cell center value and the gradient in the upwind cell and \( \Delta \vec{S} \) is the displacement vector from the upstream cell centroid to the face centroid. The gradient \( \nabla \phi \) in the cell is calculated as explained in Section 2.3.3.

2.3.3 Evaluation of Derivatives

The derivative \( \nabla \phi \) of a given variable \( \phi \) is used to discretize the convection and diffusion terms of the equations of motion. The gradient is computed using the Green-Gauss theorem as

\[
(\nabla \phi)_{c0} = \frac{1}{V} \sum_f \bar{\phi}_f A_f
\]

(2.61)

where \( \phi_f \) is the value of \( \phi \) at the cell face centroid, and the summation is over all the faces enclosing the cell.

Cell-Based Derivative Evaluation

The simplest way of finding the face value, \( \phi_f \), in Equation 2.61 is averaging the values at the neighboring cell centers (See Figure 2-4), i.e.,

\[
\bar{\phi}_f = \frac{\phi_{c0} + \phi_{c1}}{2}.
\]

(2.62)

This method is used in the cell-based evaluation.
Node-Based Derivative Evaluation

A more comprehensive approach is to use the average of the nodal values on the face

\[ \bar{\phi}_f = \frac{1}{N_f} \sum_{n} \bar{\phi}_n \]  

(2.63)

where \( N_f \) is the number of nodes on the face and \( \bar{\phi}_n \) are nodal values that are constructed from the weighted average of the cell values surrounding the nodes. The approach was originally proposed by Holmes and Connell (1989) and Rauch et al. (1991). The node-based averaging scheme is known to be more accurate than the default cell-based scheme for unstructured meshes, most notably for triangular and tetrahedral meshes (Fluent, 2006) and was used for the following work.

2.3.4 Solver

In FLUENT 6.3 two solver technologies are available, the pressure-based solver and the density-based solver. The pressure-based solver was designed for incompressible flows and was used for solving the above equations. The pressure-based solver has two solution methods, the segregated method and the coupled method. The difference between the segregated and the coupled method is the way the momentum and continuity equations are solved. In the segregated method, the momentum equations (three in 3D) and the continuity equation are solved sequentially and corrections are applied using a pressure-velocity correction method, while in the coupled method the momentum and continuity equations are solved in a coupled manner. For the present work, the segregated method was used due to its memory efficiency. The approach is explained in the following sections.

The flows modeled were all considered steady state. The measurements that the models were validated against were tow tank measurements, and even if they
had an acceleration period and a deceleration period, there existed for all of the
measurements a distinct portion of each run that appeared to be in steady state,
therefore the steady state equations were used. The integral form of the steady
state continuity and momentum equations that were used are

\[ \int \rho \vec{v} \cdot dA = 0 \] (2.64)

\[ \int \rho \vec{v} \cdot dA = \int p \mathbf{I} \cdot dA + \int \vec{\tau} dA + \int_{V} \vec{F} dV \] (2.65)

where \( \mathbf{I} \) is the identity matrix, \( \vec{\tau} \) is the stress tensor, and \( \vec{F} \) is the force vector. Otherwise the terminology is the same as in Equation 2.57.

2.3.5 Discretization of the Momentum Equation

Similar discretization as used for the scalar transport equation explained in Section 2.3.1 is used by FLUENT 6.3 for the momentum equation. The x-momentum
equation is then:

\[ a_p u = \sum_{nb} a_{nb} u_{nb} + \sum p_f A_f \cdot \hat{i} + S. \] (2.66)

The coefficients \( a_p \) and \( a_{nb} \) include mass fluxes that need to be obtained as a part of the solution and the solution is found iteratively using the AMG method explained
in Section 2.3.8.

The storage scheme used, the so-called co-located scheme, where both pressure
and velocity are stored in the cell center, requires interpolation of the pressure at
the cell face from the cell center values. There are several interpolation methods
available, but when there are jumps or large gradients in pressure such as when
working with porous media, it is advised to use the PRESTO! (PREssure STagger-
ing Option) scheme for the interpolation of pressure.

The procedure used in the PRESTO! scheme to calculate the face pressure is similar in nature to the staggered-grid schemes used with structured meshes (Patankar, 1980). A staggered control volume is defined around the face and the discrete continuity balance is used to calculate the face pressure. The approach is defined such that similar accuracy is obtained for structured and unstructured meshes (Fluent, 2006).

2.3.6 Discretization of the Continuity Equation

Discretization of Equation 2.64 employed by FLUENT 6.3 yields

\[
\sum_{f}^{N_{\text{faces}}} J_f A_f = 0 \quad (2.67)
\]

where \( J_f \) is the mass flux through face \( f \), \( J_f = \rho_f (v_n)_f \). The interpolation scheme used to find the face normal velocity, \( (v_n)_f \), is different than the ones used to find the face velocity required for the momentum equation. Those could result in unphysical checker boarding of pressure. The procedure used is similar to the one outlined by Rhie and Chow (1983) and the \( \alpha_P \) coefficient in the momentum equation (Equation 2.66) is used in a weighed averaging procedure. Using the terminology in Figure 2-4 the averaging procedure looks like

\[
J_f = \rho_f \frac{a_{p,c_0} v_{n,c_0} + a_{p,c_1} v_{n,c_1}}{a_{p,c_0} + a_{p,c_1}} + d_f ((p_{c_0} + \nabla p_{c_0}, \hat{r}_0) - (p_{c_1} + \nabla p_{c_1}, \hat{r}_1)) = \hat{J}_f + d_f (p_{c_0} - p_{c_1}) \quad (2.68)
\]

where \( p_{c_0}, p_{c_1} \) and \( v_{n,c_0}, v_{n,c_1} \) are the pressures and normal velocities, respectively, within the two cells on either side of the face, and \( \hat{J}_f \) contains the influence of velocities in these cells. The term \( d_f \) is a function of \( \bar{a}_P \), the average of the momentum
2.3.7 Pressure Velocity Coupling

The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm available in FLUENT 6.3 was chosen for the coupling between pressure and velocity. The momentum equation (Equation 2.66) is solved with a guessed pressure field (generally from the previous iteration), \( p^* \), and the face flux, \( J_f^* \), computed by Equation 2.68 is

\[
J_f^* = J_f^* + d_f(p_{c0}^* - p_{c1}^*). \tag{2.69}
\]

\( J_f^* \) does not satisfy the continuity equation (Equation 2.67) and a correction term

\[
J_f' = d_f(p'_{c0} - p'_{c1}),
\]

where \( p' \) is the cell pressure correction, is added to \( J_f^* \)

\[
J_f = J_f^* + J_f'. \tag{2.70}
\]

To obtain an equation for the pressure correction, \( p' \), Equation 2.70 is substituted into the continuity equation (Equation 2.67), which yields:

\[
ap p' = \sum_{nb} a_{nb} p'_{nb} + b \tag{2.71}
\]

where the source term \( b \) is the net flow rate into the cell:

\[
b = \sum_{f}^{N_{faces}} J_f^* A_f. \tag{2.72}
\]

The solution to Equation 2.71 is found iteratively using the AMG method (Section 2.3.8). The pressure is updated using

\[
p = p^* + \alpha p', \tag{2.73}
\]
where $\alpha_p$ is the under-relaxation factor for pressure (see Section 2.3.9), and the face flux is corrected using Equation 2.70. This corrected face flux satisfies the continuity equation.

### 2.3.8 Algebraic Multigrid

For faster convergence due to errors that vary slowly, in the spatial sense, combining the solution with a solution on a coarser grid often speeds up convergence. The multigrid method employs such a method by finding a correction to the obtained approximate solution on a coarser grid.

A set of discretized linear equations with exact solution, $\phi_e = \phi + \psi$, where $\phi$ is an approximate solution and $\psi$ is a correction to the approximate solution, is given by

\[ A\phi_e + b = 0 \]  
\[ (2.74) \]

and

\[ A\psi + (A\phi + b) = 0. \]  
\[ (2.75) \]

Before the solution is converged a defect, $d$, will be associated with the approximate solution, $\phi$

\[ A\phi + b = d. \]  
\[ (2.76) \]

Combining Equations 2.75 and 2.76 gives an equation in terms of the fine level operator, $A$, and the defect, $d$

\[ A\psi + d = 0. \]  
\[ (2.77) \]

To solve this on a coarser level requires transferring the defect to the coarse level
(restriction), solving it with a coarse level operator, and then transferring the computed correction back to the fine level (prolongation). The equations can be set up like this:

\[ A^H \psi^H + Rd = 0 \]  
\[ \phi^{\text{new}} = \phi + P\psi^H \]

where \( A^H \) is the coarse level operator, \( \psi^H \) is the coarse level correction, \( R \) is the restriction operator and \( P \) is the prolongation operator. The multigrid method used was the so-called algebraic multigrid (AMG), which had the advantage that there was no need for re-discretization on coarser levels. This was of course an advantage when using unstructured meshes. The restriction and prolongation operators were based on the additive correction strategy described for structured grids by Hutchinson and Raithby (1986) and the coarse level operator was constructed using a Galerkin approach. More information on the specifics of the multigrid method can be found in Fluent (2006).

### 2.3.9 Under Relaxation

To reduce divergent behavior of the solution to this tightly coupled set of equations, under relaxation was used. Under relaxation of variables reduces the change of the variable being solved between each iteration, by means of the under relaxation factor \( \alpha \) as follows:

\[ \phi = \phi_{\text{old}} + \alpha \Delta \phi. \]

Under relaxation of equations can be described as follows (Fluent, 2006; Ferziger and Perić, 2002):
\[ \frac{a_p \phi}{\alpha} = \sum_{nb} a_{nb} \phi_{nb} + b + \frac{1 - \alpha}{\alpha} a_p \phi_{old}. \]  

(2.81)

Under relaxation was applied as necessary. For the present work the default values in FLUENT 6.3 were used, since there were no diverging solutions.

2.4 Meshing

Meshes for modeling fluid problems using FLUENT 6.3 are created in GAMBIT, the meshing software supplied for FLUENT 6.3 (www.fluent.com). The meshes can be created on the basis of a CAD-generated geometry or, if they are simple enough, they can be built solely in Gambit which was the case for the present work. FLUENT 6.3 can handle meshes with different elements, in 2D quadrilateral and triangular elements, and in 3D hexahedral and tetrahedral elements. FLUENT 6.3 can also handle polyhedral elements, but these cannot be created in Gambit, and are created inside FLUENT 6.3 from a tetrahedral mesh.

2.4.1 Grid Generating Methods

There are two classes of grids (or meshes), structured or unstructured, and sometimes also a blend of the two is used. A structured mesh is characterized by regular connectivity that can be expressed as a two or three dimensional array. This restricts the element choices to quadrilaterals in 2D or hexahedra in 3D. The regularity of the connectivity conserves space since neighborhood relationships are defined by the storage arrangement. An unstructured mesh is characterized by irregular connectivity which is not readily expressed as a two or three dimensional array in computer memory. This allows for any possible element that a solver might be able to use. Compared to structured meshes, the storage requirements for an unstruc-
A 3D volume cell in the mesh is built up from faces that enclose the cell. The faces are enclosed by a number of edges that connect the nodes in the cell corners. A tetrahedral cell is shown in Figure 2-5. The size of a cell is defined as the length of the edges in the cell. When using unstructured meshes, the size of the elements can be controlled quite effectively by the use of size functions. The size function can prescribe the cell size as a linear function of distance from a point, edge, face or volume, either with a set starting size or from an already created edge or face mesh. The starting size can also be defined from curvature of the starting edge or face or from the distance between faces. The growth rate of the cells is defined as the increase in mesh-element edge length with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20% increase in mesh-element edge length with each succeeding layer of elements. The max size specification represents
Figure 2-5: Tetrahedral cell. The tetrahedral cell is enclosed by four triangular faces. The edges are indicated by lines and nodes are indicated by dots.

the maximum allowable mesh-element edge length for the attachment entity either inside or outside the outer boundary of the size function.

Boundary layers define the spacing of mesh node rows in regions immediately adjacent to edges and/or faces. They are used primarily to control mesh density and, thereby, to control the amount of information available from the computational model in specific regions of interest. As an example of a boundary layer application, consider a computational model that includes a cylinder representing a pipe through which flows a viscous fluid. Under normal circumstances, it is likely that the fluid velocity gradients are large in the region immediately adjacent to the pipe wall and small near the center of the pipe. By attaching a boundary layer to the face that represents the pipe wall, the mesh density can be increased near the wall and decreased near the center of the cylinder, thereby obtaining sufficient information to characterize the gradients in both regions while minimizing the total number of mesh nodes in the model. Boundary layers can be attached to edges or faces, and are defined using the height of the first row of elements and by a growth factor which specifies the thickness of each succeeding layer of elements as the thickness
of the preceding layer times the growth factor. The edge or face mesh on the edge or face that the boundary layer mesh is grown from decides the type of cells used for the boundary layer. From an edge the cells are always quadrilateral, but from a face the cells are hexahedral when the face mesh is quadrilateral and wedge shaped (extruded triangles) when the face mesh is triangular.

When using wall functions to model the region closest to the wall, it is important to keep in mind that the wall functions give the best results for $30 < y^* < 300$ (Section 2.2.4). This relation is generally used to choose the cell size next to the wall, but $y^*$ is not known before the simulation is run. Therefore the specification of the height of the first cell next to the wall often has to be an iterative approach. A height is guessed and the simulation is run, and if $y^*$ is outside the application interval of the wall functions, a new cell height needs to be specified and the simulation run again until $y^*$ is within the application interval, $30 < y^* < 300$.

2.4.2 Numerical Diffusion

The discretized equations used for the present work have some numerical diffusion. The numerical diffusion is smallest when the cells are aligned with the flow (Fluent, 2006). This is rare, but can happen, for example when modeling flow through a long narrow channel using quadrilateral elements, or hexahedral elements in 3D. Triangular elements, or tetrahedral elements in 3D, however are never aligned with the flow and tend to have more numerical diffusion than quadrilateral or hexahedral. The main reason for this difference between the element types is due to the interpolation process for finding variable values at cell faces from the values at cell centers.
2.4.3 Refinement of the Mesh

For computational fluid dynamics (CFD) simulations to give reliable results, it is necessary to use a mesh that can resolve the fluid dynamic problem in sufficient detail. One statement of when the flow is described in sufficient detail is that any flow feature should be described with at least 5 cells, preferably more (Fluent, 2006).

One way to figure out when the mesh is refined enough, is to start out with a coarse mesh and then refine it until the results of the simulations do not change any more, but this approach is not always applicable, e.g. due to lack of computational power, and thus a more subjective approach has to be used where the user decides which features need to be resolved.

2.5 Using the Model

The geometries in the following work were described using volume cells, tetrahedral, hexahedral and prism cells, and the equations were solved using a steady state finite volume approach using an iterative method. The equations used for the finite volume approach were the steady state volume integrated versions of Equations 2.6 and 2.8 discretized as explained in Section 2.3. The steady state problems were solved using an iterative method where a solution was guessed and iterated until convergence using an under relaxation iteration approach.

The input required to run the model is a mesh, a set of boundary conditions and a specification of which method of solutions is to be used. When the porous media model is used, the porous media resistance coefficients are also needed as input and have to be found from a theoretical relationship or from measurements. In FLUENT 6.3, two methods are available to calculate the elements in the porous media resistance matrices ($D_{ij}$ and $C_{ij}$) for a volume. When porous media re-
sistance coefficients are given in 3 orthogonal directions and vectors defining the directions, the matrices containing the porous media resistance coefficients in the global reference system are defined using the method explained in Section 3.3. The resistance matrices can also be defined in a conical manner where the tangential pressure drop coefficients are defined along the vertical and horizontal tangents of the cone and the normal porous media resistance coefficients are defined normal to the cone (Fluent, 2006).

When a turbulence model is used, the turbulent quantities in the incoming water need to be specified at the inlet boundaries. For the k-ε models these are the specification of k and ε. The level of turbulence at the inlet boundaries can influence the solution, generally by adding more diffusion, and testing the influence of different quantities is advised.

The basic outputs are fluid velocity and pressure distributions, but a number of other outputs are available. Examples are: drag on solid objects, distribution of turbulence if a turbulence model is used and concentrations if they are modeled.

The postprocessing software in FLUENT 6.3 does not have an option for showing drag- or lift force on a porous media. Therefore the forces on the net panel had to be found using some other method. Control volume calculations can be useful for such problems, and since FLUENT 6.3 is capable of calculating surface integrals of all the variables in Equations 2.91 and 2.93, these equations were used to find the forces on the net panel. When the frame around the net panel was included in the model (see Sections 3.1 and 3.2.3 for more information on the setup), the control volume either had to enclose the net panel very closely, and not enclose the frame around the net panel, or alternatively enclose both the net and the frame. If the control volume enclosed both the net panel and the frame, the drag force on the net panel was found by subtracting the drag on the frame from a simulation with no porous media from the drag on the frame and net panel. This is the same approach
that was used when making tank measurements of the drag on net panels.

### 2.6 Control Volumes

The linear momentum equation for a control volume can be written as the time rate of change of momentum equals the sum of the forces (White, 1994):

\[
\frac{d}{dt}(m\mathbf{u}) = \sum \mathbf{F} = \frac{d}{dt}(\int_{CV} \mathbf{u}\rho dV) + \int_{CS} \mathbf{u}\rho (\mathbf{u}_r \cdot \mathbf{n})dA \tag{2.82}
\]

where \( \mathbf{u} \) is current velocity, \( \mathbf{u}_r \) is current velocity relative to the reference frame (if the reference frame is stationary \( \mathbf{u}_r = \mathbf{u} \)) and \( \mathbf{n} \) is a unit vector pointing out of the control volume at a right angle to the control volume surface. Subscripts CV and CS specify integration over the control volume and over the surface of the control volume respectively.

The forces acting on a control volume are for example forces from normal stresses (pressure), \( F_p \),

\[
F_p = \int_{CS} \rho (-\mathbf{n})dA \tag{2.83}
\]

shear stresses, \( F_s \), or external forces, \( F_{Ext} \), on some object in the control volume.

The momentum equation as shown in Equation 2.82 is a vector equation and for a steady state problem in a stationary reference frame the x-component is:

\[
\sum F_x = \int_{CS} u\rho (\mathbf{u} \cdot \mathbf{n})dA \tag{2.84}
\]

and the forces can be due to pressure, \( p \), shear stress, \( \tau \), and due to some external force in the x-direction, \( F_{Ext,x} \),

\[
\sum F_x = \int_{CS} p(-n_x)dA + \int_{CS} \tau_x dA + F_{Ext,x} \tag{2.85}
\]
Figure 2-6: Control volume used to calculate forces on a net panel in current. The net panel needs to be positioned inside the control volume. The labels $x$ – neg, $x$ – pos etc. are defining the different faces of the control volume and $u$ is the incoming current.

where $\tau_x$ is the x-component of the shear stresses.

If the flow is laminar the shear stresses are calculated as

$$\tau_{xy} = \mu \frac{\partial u}{\partial y} \quad (2.86)$$

$$\tau_{xz} = \mu \frac{\partial u}{\partial z} \quad (2.87)$$

where $\mu$ is the molecular viscosity of the fluid, and if the flow is turbulent they are calculated as

$$\tau_{xy} = \mu_{eff} \frac{\partial u}{\partial y} \quad (2.88)$$

$$\tau_{xz} = \mu_{eff} \frac{\partial u}{\partial z} \quad (2.89)$$

where $\mu_{eff} = \mu + \mu_t$ is the effective viscosity that is the sum of the laminar viscosity and the eddy viscosity.

Equations 2.84 and 2.85 can be set up in a simplified form for a right angled hexahedral control volume oriented along $x$-, $y$- and $z$-axes (Figure 2-6). The $x$-
The component of the forces is

\[
\sum F_x = -F_{\text{ext}_x} + \int_{x-\text{neg}} pdA - \int_{x-\text{pos}} pdA \\
- \int_{y-\text{neg}} \tau_{xy} dA + \int_{y-\text{pos}} \tau_{xy} dA - \int_{z-\text{neg}} \tau_{xz} dA + \int_{z-\text{pos}} \tau_{xz} dA \quad (2.90)
\]

and the x-component of the right hand side of Equation 2.84 is

\[
\int_{CS} u\rho(\mathbf{u} \cdot \mathbf{n}) dA = \int_{x-\text{neg}} u\rho(-u) dA + \int_{x-\text{pos}} u\rho dA + \int_{y-\text{neg}} u\rho(-v) dA \\
+ \int_{y-\text{pos}} u\rho dA + \int_{z-\text{neg}} u\rho(-w) dA + \int_{z-\text{pos}} u\rho dA. \quad (2.91)
\]

For the y-component the equations are

\[
\sum F_y = -F_{\text{ext}_y} + \int_{y-\text{neg}} pdA - \int_{y-\text{pos}} pdA \\
- \int_{x-\text{neg}} \tau_{yz} dA + \int_{x-\text{pos}} \tau_{yz} dA - \int_{z-\text{neg}} \tau_{yz} dA + \int_{z-\text{pos}} \tau_{yz} dA \quad (2.92)
\]

and

\[
\int_{CS} v\rho(\mathbf{u} \cdot \mathbf{n}) dA = \int_{x-\text{neg}} v\rho(-u) dA + \int_{x-\text{pos}} v\rho dA + \int_{y-\text{neg}} v\rho(-v) dA \\
+ \int_{y-\text{pos}} v\rho dA + \int_{z-\text{neg}} v\rho(-w) dA + \int_{z-\text{pos}} v\rho dA. \quad (2.93)
\]

The external forces \((F_{\text{ext}_x} \text{ and } F_{\text{ext}_y})\) can be found by isolating these terms if the rest of the terms are known. The use of these equations is twofold in the presented work:

1. The entire equation was used as the accurate calculation of the drag force and the lift force on a porous region in a CFD domain. The control volume had to enclose the entire porous media and be oriented along the coordinate
axes. The porous media did not have to fill the control volume and could be oriented arbitrarily.

2. A simplified version that was only valid for a porous media with a thickness much smaller than the other dimensions was used in the method for finding the porous media resistance coefficients. Here it was assumed that the only important terms were the external force and the pressure terms. This version used a control volume that was the same as the volume of porous media, and was of course not valid if the net panel was oriented parallel to the flow or at very small angles of attack. Making the additional simplification that the pressures on the faces of the thin porous media are uniform gave the following momentum balance

\[ F_x = (\Delta p)_x A_x \]  

(2.94)

where \((\Delta p)_x\) is the average pressure drop through the porous media in the x-direction and \(A_x\) is the projected area of the porous media in the x-direction. The same was applied to the y-direction when the net panel was not normal to the flow.
CHAPTER 3

NET PANEL STUDIES

When testing a computer model used to describe flow through a net, it is important to have laboratory measurements of some basic shape to "calibrate" the model, and to demonstrate the capabilities of the model. The basic shape used mostly by researchers is a flat net panel generally stretched on a stiff frame (Løland, 1991; Le Bris and Marichal, 1998; Zhan et al., 2006). The net panel can be set up in a tow tank and towed at different speeds and oriented with different angles of attack. A sketch of a general tow tank setup for measuring drag and lift forces on a net panel oriented at different angles of attack was shown in Figure 1-1. Measuring current reduction behind the net panel and drag and lift force on the net panel as a function of tow speed and angle of attack gave a good basis for evaluating the capabilities of the model to simulate the flow through and around a net.

Some work by Patursson et al. (2006), using CFD to model the flow through net panels at different angles of attack, has already been compared to published data by Løland (1991) and Le Bris and Marichal (1998). This previous effort indicated that the porous media model looked like a promising model to simulate the flow through nets, but more work was needed, especially to calibrate the model such that flow through the net at small angle of attack was modeled correctly. The work presented in the present Chapter develops a methodology for this calibration.

The tank and net panel setup was modeled in 3D using the CFD model. As is common in CFD modeling, a sensitivity analysis for the refinement of the mesh and for applying different model settings was performed. Because CFD runs are
time consuming, an analytical method was derived to optimize the porous resistance coefficients from measurements of drag and lift force as a function of angle of attack and tow speed. The method included an analytical description of the drag- and lift force on the net panel, applying the equations for flow through porous media and appropriate approximations. Using a fitting procedure, the porous resistance coefficients that minimize the error between measured and calculated drag and lift force were found. Using the best-fit porous resistance coefficients a comparison was made between the measured drag and lift force and current reduction and the predictions using the CFD approach. The porous coefficients were also varied up and down to check the correctness of the analytical method for finding the porous resistance coefficients.

Based on the available data (Rudi et al., 1988; Aarsnes et al., 1989; Zhan et al., 2006) and a time constraint, it was chosen to perform a series of measurements at different angles of attack on only one net panel. The measurements included drag and lift force measurements and measurements of the current in different positions behind the net. The tests were done at four different speeds and seven different angles of attack. The main objective was to get a solid set of current measurements behind the net panel as a function of angle of attack, which was lacking in the available data. The net panel chosen was a knotless nylon net used by the aquaculture industry in the Faroe Islands, with a solidity of 0.20. It would have been interesting to include a net with higher solidity in the measurement series, to assess the effect of solidity. Most aquaculture nets deployed in the ocean have some amount of biofouling attached to them which increases the solidity. This was to some extent compensated for by applying the method for finding porous resistance coefficients to the drag and lift force data by Rudi et al. (1988), but there was no current measurement data available for the high solidity net in this data series.
The CFD model was calibrated against the drag force and the lift force, while the current reduction measurements were used for a first simple validation. It was chosen to only model the square frame made of 26.7mm outside diameter stainless steel pipe around the net panel and not the rest of the supports, since these were not thought to influence the measurements significantly. The tow tank and net panel setup was modeled as a stationary system with water flowing through the tank instead of the net moving. The tank cross-section dimensions were used, and the net was positioned 5m away from the inlet boundary to give room for the stagnation in front of the net and plenty of room for the wake to stabilize.

3.1 Measurements

A series of measurements of drag and lift force on a net panel and velocity reduction behind the net panel were conducted in the UNH 37m (120ft) long, 3.66m (12ft) wide and 2.44m (8ft) deep tow/wave tank as described by Patursson (2007). The test setup shown in Figure 3-1 was constructed and set up in the tank to support the net panel and measure the forces. Drag force and lift force were measured using a load cell arrangement. Current was measured using an acoustic doppler velocimeter (ADV) in different positions behind the net, mostly in the wake region, but also outside the wake. The velocity of the carriage was measured using a set of light gates measuring travel time over a set distance.

3.1.1 Measurement Setup

The net panel was positioned well below the water surface to minimize drag forces from surface wave creation due to the pressure difference between the front and the back of the net panel. Based on preliminary CFD investigations, it was decided to position the net panel in the center of the cross-section of the tank. The CFD
investigations indicated that the wave height of the surface wave created by the net in this position would be on the order of a few millimeters and was considered negligible. The setup was designed such that the structure deformation was small and the angle of attack of the net panel could be set within ±2° (±1° from setting the angle of attack and ±1° from bending of the structure).

The basic components were a frame that holds the net panel, which was attached to a strut, and a loadcell arrangement that was attached below the carriage close to the water surface. Figure 3-1 shows the setup, along with dimensions and material description, and a picture of the setup can be seen in Figure 3-2. The strut passed through the loadcell arrangement, and at the upper end it was attached to a universal joint that allowed bending but no swiveling. The strut could be oriented arbitrarily around the vertical axis, and the angle of attack of the net panel could be set using a protractor on top of the universal joint.

The attachment of the ADV to the tow carriage was made using I-beams, C-
beams and clamps as shown in Figure 3-2. The ADV could be positioned with an accuracy of ±2cm using this setup.

The loadcell arrangement was built on top of a 1in aluminum plate, where the two loadcells were positioned perpendicular to each other, one in the drag direction and the other in the lift direction as shown in Figure 3-3. A collar was attached to the strut at the loadcell level which acted as the connection point between the strut and the loadcells. The loadcells were attached to the collar at one end and the plate at the other using pinned joints.

The loadcell arrangement was calibrated in place by pulling from the center of the net panel and measuring the force with a calibrated loadcell. A slightly nonlinear calibration curve was found. The accuracy of the force measurement was found to be good, ±0.5N, but the accuracy of the angle setting, ±2°, was of greater importance.
Figure 3-3: Topview of the loadcell arrangement. The dimensions are in inches.

Figure 3-4: Instrumentation for measuring carriage speed and current velocity. On the left figure the light gates for measuring carriage speed can be seen positioned along the tank rail, and on the figure to the right a picture of the ADV used for measuring the current velocity is shown.
Table 3.1: Accuracy of ADV measurements according to data from the supplier. (Vel. range is the velocity range setting of the instrument)

<table>
<thead>
<tr>
<th>Vel. range (cm s^-1)</th>
<th>Vel. range (cm s^-1)</th>
<th>e_{dat} (cm s^-1)</th>
<th>e_{dat} (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>12.5</td>
<td>10</td>
<td>0.35</td>
<td>2.80</td>
</tr>
<tr>
<td>25</td>
<td>10</td>
<td>0.35</td>
<td>1.40</td>
</tr>
<tr>
<td>50</td>
<td>30</td>
<td>0.55</td>
<td>1.10</td>
</tr>
<tr>
<td>75</td>
<td>100</td>
<td>1.25</td>
<td>1.66</td>
</tr>
</tbody>
</table>

The speed of the tow carriage was measured using a set of light gates (see Figure 3-4) that measured the travel time of the carriage over a set distance. The distance used for these measurements was 1.395m ±0.002m, and the measurement was made with 1ms (millisecond) accuracy.

The current velocity was measured using the ADV shown in Figure 3-4. The ADV used is a downlooking 10MHz ADVField Probe with optional sensors, connected to an ADVField processor, all from SonTek/YSI, Inc., with datalogging on a computer on the tow carriage. The accuracy for the ADV at the carriage speeds used, obtained from the supplier’s specs, are given in Table 3.1.

The speed measured by the ADV without the net panel in place and the carriage speed measured by the light gates were compared, and a slightly nonlinear relationship was found that did not fall within the accuracy specified in Table 3.1. A calibration curve was found that was applied to the ADV data to make up for this error. After calibration, all data was well within the specified error.

The tested net (shown in Figure 3-5) was a 1m by 1m knotless nylon net with twine diameter \( d = 2.8\text{mm} \) \((210d/108)\) and mesh size (length of one mesh bar between adjacent knot centers) \( \lambda = 29\text{mm} \) when it was stretched onto the frame. The net is common in salmon farming cages in the Faroe Islands and is sold as a 25mm mesh size by Refa Froystad Group AS. The 25mm dimension was probably measured inside the mesh and not center to center of the knots and without as much stretching force as applied here.
Net solidity can be defined as twine projected area divided by outline area and evaluated according to Løland (1991) as

\[ S = \frac{2d}{\lambda} + \frac{1}{2} \left( \frac{d}{\lambda} \right)^2. \]  

For the net used, solidity \( S \) was estimated to be 0.198. The solidity was also found using a digital photographic method where the pixels of the background color can be sorted from the pixels of the net color (Figure 3-5, left). Using this technique, the solidity \( (S) \) was estimated to be 0.20 when the net was stretched on the frame.

The measurements were performed at the tow speeds 12.5 cm s\(^{-1}\), 25 cm s\(^{-1}\), 50 cm s\(^{-1}\) and 75 cm s\(^{-1}\) and at the angles of attack 0°, 15°, 30°, 45°, 60° and 90°. In addition one test was done at 50 cm s\(^{-1}\) and 75°.
3.1.2 Dataprocessing

Drag and lift forces acting on the empty frame and the frame with the net were found separately using loadcell data. The data was processed using the relationship between output voltage and force found during calibration, and time series of the forces were found as illustrated in Figure 3-6. One value for each of the forces was found for each run as an average of a chosen section of the time series. The section used for averaging was chosen as the later part of the tow tank run after the startup transients evened out and before deceleration of the carriage. The zero voltage was used in the processing to re-zero the force calibration and was found for each run as an average of the part of the time series that was collected before the run started. All drag and lift force measurements were represented by at least two measurements on the empty frame and at least one measurement on the frame with the net.

The drag force \( D \) and lift force \( L \) on the net was found by subtracting the averaged measurements for each speed setting and angle of attack on the frame from the averaged measurements for the corresponding speed setting and angle of attack on the frame and net:

\[
D = \langle D_{\text{net+frame}} \rangle - \langle D_{\text{frame}} \rangle
\]

(3.2)

and

\[
L = \langle L_{\text{net+frame}} \rangle - \langle L_{\text{frame}} \rangle.
\]

(3.3)

The drag coefficient \( C_d \) and lift coefficient \( C_l \) of the net panel at each speed and angle of attack was calculated as

\[
C_d = \frac{D}{\frac{1}{2} \rho A(u_0)^2}
\]

(3.4)
Figure 3-6: Time series of drag force, lift force and \( u \) and \( v \) components of measured current velocity. The section of the time series for calculating average values is between the black lines.

\[
C_l = \frac{L}{\frac{1}{2} \rho A (u_0)^2}
\]  

(3.5)

where \( A \) is the outline area of the net panel (1m\(^2\)) and \( \langle u_0 \rangle \) is the average tow speed for all measurements at the same speed setting and angle of attack measured by the light gates.

The velocity reduction \( U_r \) in chosen locations was found by

\[
U_r = \frac{u_0 - \langle u \rangle}{u_0}
\]  

(3.6)

where \( u_0 \) is the current velocity measured using the light gates and \( \langle u \rangle \) is the time series average of the current measured by the ADV with the frame and net panel in place. The velocities \( u_0 \) and \( \langle u \rangle \) were from the same run.
3.1.3 Results and Discussion

A plot of drag and lift coefficients ($C_d$ and $C_l$) as function of angle of attack ($\alpha$) for the different velocities used is shown in Figure 3-7. The plot shows a clear trend for both drag and lift coefficients with angle of attack and that, for the most part, the drag and lift coefficients decreased with increasing speed, but it should be noted that the measurement error was quite large compared to the observed variation with speed. The lift coefficient was negative at $\alpha = 90^\circ$, and although this seemed strange, it could have been caused by some asymmetric feature of the net itself. Variations of $\alpha$ showed that $\alpha$ needed to be quite large and well outside the accuracy of the angle setting to obtain a zero lift force.

A plot of the current reduction ($U_R$) as a function of $\alpha$ for the different velocities used is shown in Figure 3-8. The plot shows that for most of the speeds the current...
Figure 3-8: Current reduction behind the net panel. The figure on the left shows current reduction as a function of angle of attack and carriage speed, measured 2.5m behind the center of the net panel. The figure on the right shows current reduction as a function of angle of attack and distance across the tank from the centerline 2.5m behind the net panel. The dashed lines show the position of the vertical frame post for the different angles of attack. All measurements on the figure to the right were performed at 0.5m s\(^{-1}\).

Current reduction behind the center of the net panel was of similar magnitude from \(\alpha = 90^\circ\) to \(\alpha = 60^\circ\), and became larger for lower \(\alpha\), but for \(\alpha = 0^\circ\), the current reduction is smaller than for \(\alpha = 15^\circ\). This could be due to the fact that for \(\alpha = 0^\circ\) the main contribution to the current reduction was the front vertical pipe of the frame, while the rest (net panel and rear vertical pipe) was hidden in the wake of the front pipe and affecting the flow less than at a small, but larger than zero, angle of attack.

The variation with speed was quite small compared to the associated measurement error, but it is interesting to see that the current reduction was generally smaller at 12.5cm s\(^{-1}\) than at 25cm s\(^{-1}\), while in Figure 3-7 it can be seen that the drag coefficient was generally larger at 12.5cm s\(^{-1}\) than at 25cm s\(^{-1}\). This might have to do with increased mixing in the wake at slower speeds. It is also interesting to note that the low drag coefficient at \(u = 12.5\text{cm s}^{-1}\) and \(\alpha = 90^\circ\) is reflected in a low current reduction.
Figure 3-8 also shows a few measurements of velocity reduction as a function of angle of attack and distance across the tank from the centerline 2.5m behind the net panel. Here it can be seen that the wake from the frame posts reached at least 2.5m behind the frame. It can also be seen that there was a slight increase in velocity (negative reduction) outside of the wake.

During the tows, a quite significant vibration of the frame was observed, which might have led to errors in the force measurements. At low speeds the carriage movement was unsteady and vibrating which forced a vibration of the frame, and at high speeds the vortex shedding from the vertical parts of the frame and frame supports forced a quite powerful vibration of the frame. The measurements at $u_0 = 50 \text{cm s}^{-1}$ seemed to be least affected by the vibrations. The vibrations resulted in fluctuating force measurements, but since the data was presented as average values of long time series, the accuracy of the data was assumed reasonable, but it is not easy to give an accurate error analysis. Most of the data points included some kind of replicate measurement, and taking the standard deviation of these measurements gave some kind of error estimate. For the drag measurement most of the standard deviations were within 5% of the measured value, while some are higher, especially those at $\alpha = 0^\circ$. For the lift measurement, most of the standard deviations were within 10% of the measured value, but some were much higher. For $\alpha = 45^\circ$ and $u_0 = 75 \text{cm s}^{-1}$ the standard deviation for the lift measurement was over 40% of the measured value. Overall the measurements at $u_0 = 50 \text{cm s}^{-1}$ seemed to have the highest accuracy, and the drag data seemed to be more accurate than the lift data.
3.2 Preliminary Computational Fluid Dynamic Runs

To determine what settings and computational mesh to use when running the CFD model, a few preliminary runs were made. These were run before and in parallel to the development of the methodology to assess the porous resistance coefficients, so coefficients varied as progress was made.

3.2.1 Testing Dependence on Thickness of Porous Media

A few simulations were run to test the influence of the thickness of the porous media on flow through the net panel. The test was performed on a net panel 1m by 1m oriented normal to the flow ($\alpha = 90^\circ$) and with a relatively small angle of attack ($\alpha = 30^\circ$). The net panel was centered in the cross-section of a rectangular channel enclosed by frictionless walls (width = 4m, depth = 2.5m, length = 20m) 5m downcurrent from the inlet. Three thicknesses were tested - 5, 10 and 50mm. The thicknesses were chosen to be comparable to or larger than the thickness of the net, but much smaller than the size of the net panel. The nets were modeled without the structural support, such that it was only the thin porous media immersed in a fluid domain. First the exact same mesh was used for all 3 thicknesses, but the porous region occupied regions corresponding to the different thicknesses, and resistance was only added to the appropriate part. The magnitude of the resistance coefficients was inversely proportional to the thickness of the porous media to achieve the same pressure drop, at normal angle of attack, through porous media of different thickness. The resistance normal to the net panel was chosen from a curve fit between drag force data on a net panel at normal angle of attack and current velocity through the net, and the resistance tangential to the net was set to 31% of the resistance normal to the net. The resistance coefficients used are listed in Table 3.2.
Table 3.2: Porous media resistance coefficients associated with the different thickness of porous media.

<table>
<thead>
<tr>
<th>Thickness (mm)</th>
<th>$D_n$ (m$^{-1}$)</th>
<th>$D_i$ (m$^{-1}$)</th>
<th>$C_n$ (m$^{-2}$)</th>
<th>$C_t$ (m$^{-2}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>$340.0 \times 10^3$</td>
<td>$105.4 \times 10^3$</td>
<td>57.80</td>
<td>17.92</td>
</tr>
<tr>
<td>10</td>
<td>$170.0 \times 10^3$</td>
<td>$52.70 \times 10^3$</td>
<td>28.90</td>
<td>8.96</td>
</tr>
<tr>
<td>50</td>
<td>$34.00 \times 10^3$</td>
<td>$10.54 \times 10^3$</td>
<td>5.78</td>
<td>1.79</td>
</tr>
<tr>
<td>50*</td>
<td>$34.00 \times 10^3$</td>
<td>$10.54 \times 10^3$</td>
<td>5.78</td>
<td>1.79</td>
</tr>
</tbody>
</table>

* Coarse mesh

The porous media was meshed using cubic elements with 5mm sides, and the rest of the domain was modeled using tetrahedral elements increasing in volume with distance away from the porous media up to a maximum size of 150mm. The growth rate was set to 1.2. This gave a mesh that was fine inside and close to the porous media, but coarsening quite quickly with distance away from the porous media. Less than 1m away from the porous media, the cells were at their maximum size.

Second a coarse mesh with a 50mm thick porous media at $\alpha = 30^\circ$ was created, where the edge of the porous media was modeled using a 5mm cell size, (tetrahedral cells) but with the cell size increasing away from the edge of the porous media, both inside and outside the porous media. Growth rate was 1.2, and maximum cell size inside the porous media was 50mm, and outside the porous media it was 150mm. The use of frictionless walls meant that the shear forces on the walls of the tank and towards the surface were small, and the large cell size at the boundaries was not considered a problem.

Results from simulation runs are provided in Table 3.3. From these simulations it seems like the thickness of the porous media does not influence the simulations significantly.
Table 3.3: Comparing drag- and lift force on the net panel and velocity reduction 2.5m behind the center of the net panel.

<table>
<thead>
<tr>
<th>Thickness (mm)</th>
<th>(90^\circ)</th>
<th>(30^\circ)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(D (N))</td>
<td>(L (N))</td>
</tr>
<tr>
<td>5</td>
<td>32.53</td>
<td>0</td>
</tr>
<tr>
<td>10</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>50</td>
<td>32.55</td>
<td>0</td>
</tr>
<tr>
<td>50*</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Thickness (mm)</th>
<th>(C_d)</th>
<th>(C_l)</th>
<th>(U_R)</th>
<th>(C_d)</th>
<th>(C_l)</th>
<th>(U_R)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0.2602</td>
<td>0</td>
<td>0.124</td>
<td>0.1188</td>
<td>0.0744</td>
<td>0.114</td>
</tr>
<tr>
<td>10</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>0.1227</td>
<td>0.0747</td>
<td>0.118</td>
</tr>
<tr>
<td>50</td>
<td>0.2604</td>
<td>0</td>
<td>0.124</td>
<td>0.1245</td>
<td>0.0751</td>
<td>0.120</td>
</tr>
<tr>
<td>50*</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>0.1232</td>
<td>0.0748</td>
<td>0.116</td>
</tr>
</tbody>
</table>

* Coarse mesh

Table 3.4: Different cases tested.

<table>
<thead>
<tr>
<th>Case</th>
<th>Wall function</th>
<th>Wall modeling</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>Standard wall functions</td>
<td>Moving wall</td>
</tr>
<tr>
<td>3</td>
<td>Non-equilibrium wall functions</td>
<td>Moving wall</td>
</tr>
<tr>
<td>4</td>
<td>Enhanced wall treatment</td>
<td>Moving wall</td>
</tr>
<tr>
<td>5</td>
<td>Enhanced wall treatment</td>
<td>Zero shear force</td>
</tr>
</tbody>
</table>

3.2.2 Testing Dependence on Models Used in FLUENT

To test what settings or models in FLUENT 6.3 give the best results, and therefore would be used for running subsequent simulations, different combinations of these models were tested. The models investigated included different near wall treatments, different ways of modeling the walls of the tank, different turbulence models and a laminar model for the porous media region. Different cases were set up where the different models were tested on the same mesh. Some of the different cases tested are identified in Tables 3.4 and 3.5.

The mesh used for testing these models was built using the real dimensions of

Table 3.5: Different cases tested.

<table>
<thead>
<tr>
<th>Case</th>
<th>Flow in porous media</th>
<th>Wall modeling</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>Laminar</td>
<td>Zero shear force</td>
</tr>
</tbody>
</table>
Table 3.6: Porous coefficients used.

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D_n$</td>
<td>68370m$^{-2}$</td>
</tr>
<tr>
<td>$D_t$</td>
<td>45060m$^{-2}$</td>
</tr>
<tr>
<td>$C_n$</td>
<td>5.037m$^{-1}$</td>
</tr>
<tr>
<td>$C_t$</td>
<td>1.511m$^{-1}$</td>
</tr>
</tbody>
</table>

Table 3.7: Comparing results from the different cases. Listed are drag ($D$) and drag coefficient ($C_d$) on the frame and current 2.5m behind the net panel at the centerline of the net panel ($u$) and the minimum current ($Min(u_{across})$) in the wake of the frame also 2.5m behind the panel.

<table>
<thead>
<tr>
<th>Case</th>
<th>$D$ (N)</th>
<th>$C_d$</th>
<th>$u$ (cm s$^{-1}$)</th>
<th>$Min(u_{across})$ (cm s$^{-1}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>14.41</td>
<td>1.051</td>
<td>44.9</td>
<td>41.9</td>
</tr>
<tr>
<td>3</td>
<td>14.41</td>
<td>1.051</td>
<td>44.9</td>
<td>41.9</td>
</tr>
<tr>
<td>4</td>
<td>15.03</td>
<td>1.097</td>
<td>44.9</td>
<td>41.8</td>
</tr>
<tr>
<td>5</td>
<td>15.03</td>
<td>1.097</td>
<td>44.9</td>
<td>41.8</td>
</tr>
<tr>
<td>7</td>
<td>14.34</td>
<td>1.046</td>
<td>44.9</td>
<td>42.0</td>
</tr>
</tbody>
</table>

the tow tank with the net panel and frame. The net panel had dimensions 1m by 1m and was modeled using a 10mm thick porous media. The porous media was meshed using 5mm by 10mm by 10mm hexahedral cells, such that the thickness of the porous media was comprised of 2 cells. The frame pipe had a 3 layer boundary layer mesh using 10mm by 10mm quad cells with a 3mm thickness in the first layer and a growth factor of 0.95 for each layer away from the pipe wall. From the centerline of the pipes making up the frame to 2.5m behind the frame, the cell size was kept at 25mm. The rest of the domain was meshed using the TGrid scheme with a growth rate of 1.2 except in the wake of the frame around the net panel where the growth rate was 1.1 from the set cell size behind the frame. The max cell size in the domain was set to 150mm. No restriction was made on the cell size at the domain boundaries, which for the most part was 150mm. All simulations were run using an incoming velocity of 0.5m s$^{-1}$, and $k = 0$ and $\epsilon = 0$ at the inlet boundary. The porous resistance coefficients used are listed in Table 3.6.

The simulation results so far are summarized in Table 3.7. It is interesting to
note that for the present mesh, there was no significant difference between using different kinds of near wall treatments and, more importantly, the two different ways of describing the wall. The water surface of the tank was described as a wall with zero shear force, and as the mentioned results show, the sides and the bottom of the tank can also be described this way. Using the laminar model for the porous media region did not affect the results significantly.

Using these findings and the fact that the net panel was vertically centered in the tank, there was a horizontal symmetry plane. Thus, there was a possibility of modeling only half the domain above or below the symmetry plane, giving possibility for a finer mesh. This option was explored further by running simulations of only the upper half of the domain defining the face towards the other half of the domain as a symmetry plane. The symmetry plane seemed to affect the results such that the flow was more diffusive along the symmetry plane than in the rest of the domain, and this option was not further explored.

For simulations of the net panel at small angles of attack, when looking at the velocity reduction, it was important to model the wake of the frame around the net panel with reasonable accuracy. The first requirement to achieve that was to model the drag force on the frame with good enough accuracy. The way the measurements of drag on the empty frame during the tank tests were performed and the fact that they included drag on other supports that were not included in the CFD model, made these measurements improper for a validation. Therefore the drag coefficient based on the projected area of the frame was calculated from the drag force obtained from the CFD-results. The calculated drag coefficient was close to the expected drag coefficient of a slender cylinder \((C_d = 0.98\) for a cylinder with length to width ratio of 40 at \(Re \geq 10^4\) (Edward J. Shaughnessy et al., 2005)) or maybe a little bit higher, which could have been due to the added pressure drop due to the presence of the net panel. Using the enhanced wall treatment did produce a higher \(C_d\) and
did not seem to be an improvement over the standard wall function in this case.

The dependence of using different $k$-$\epsilon$ turbulence models for 2D flow around net panels described as porous media was investigated by Patursson et al. (2006), and there it seemed like the realizable $k$-$\epsilon$ (RKE) model was best suited for this problem. The main difference between the results using the standard $k - \epsilon$ (SKE) and the RKE models was a generally more diffusive solution by the SKE model and a more stable solution by the RKE model, but the difference was small. The RKE model even seemed to converge faster in some cases.

### 3.2.3 CFD-Mesh used for Net Panel and Frame

For simulations of the flow through and around the net panel, it was important to model the wake of the frame around the net panel with reasonable accuracy since this acted as a border between the reduced velocity behind the net panel and the rest of the flow. The small diameter of the pipe used for the frame necessitated a fine mesh around the pipe and behind it. This was achieved by defining four planar surfaces that extended 3m downstream behind each member making up the frame. When the net panel was located parallel to the flow, this reduced to one vertical surface inside the net panel and behind the frame. The mesh was grown from the frame and these surfaces with a preset start size, a growth factor and preset maximum size.

A number of meshes were tested to model the full domain with the net and frame (Table 3.8). The net was modeled as a 5cm thick porous media. The meshes were created using the Gambit software, and two meshing methods were used - TGrid and Hex Core. The Hex Core meshing option created meshes with significantly lower number of cells for the same spatial resolution, and the hexahedral cells, that were aligned quite well with the flow, had lower numerical diffusion than tetrahedral cells. That made the Hex Core meshing routine promising.
Table 3.8: Different cases tested modeling the full domain using a 50mm thick porous media.

<table>
<thead>
<tr>
<th>Case</th>
<th>Mesh type</th>
<th>Boundary Layer</th>
<th>Cell size around frame</th>
<th>Cell size in wake</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Size (mm)</td>
<td>G.F.</td>
</tr>
<tr>
<td>71</td>
<td>TGrid</td>
<td>Hexahedral</td>
<td>10</td>
<td>1.2</td>
</tr>
<tr>
<td>72</td>
<td>Hex Core</td>
<td>Hexahedral</td>
<td>10</td>
<td>1.2</td>
</tr>
<tr>
<td>73</td>
<td>Hex Core</td>
<td>Wedge</td>
<td>5</td>
<td>1.2</td>
</tr>
<tr>
<td>74</td>
<td>Hex Core</td>
<td>Wedge</td>
<td>10</td>
<td>1.2</td>
</tr>
<tr>
<td>76</td>
<td>Hex Core</td>
<td>Hexahedral</td>
<td>10</td>
<td>1.2</td>
</tr>
</tbody>
</table>

It proved very important to have a boundary layer mesh enclosing the frame around the net panel. Without this boundary layer mesh, the boundary layer around the frame became unstable with highly varying strain rates resulting in a highly varying wake structure. The best results were obtained using a boundary layer built by extruding layers of hexahedral cells from a surface mesh of quadrilateral cells. Boundary layers made by extruding layers of wedge cells from a surface mesh of triangular cells also worked, but did not give as smooth a solution as the ones out of hexahedral cells.

The difference in using meshes from the TGrid and Hex Core meshing schemes was mostly in the diffusion of the wake from the frame and the number of cells used. The computational power available limited the number of cells in the mesh to around $4 \cdot 10^6$ cells, in order to achieve a steady state solution within reasonable time (24hr). This limit made it necessary to use a rather coarse mesh away from features that were important to resolve. These features included areas with large gradients. The minimum possible cell size in the wake of the frame using the Tgrid meshing scheme was around 20mm, while a similar mesh using the Hex Core meshing scheme could be created with 10mm mesh size in the wake of the frame with less than half the number of cells.

A number of meshes were tested, and velocity reduction across the wake of the net panel from a few are shown in Figure 3-9. The figure gives insight into the most
important difference between the meshes. The Hex Core meshes created a narrower wake behind the frame than the TGrid mesh, and the boundary layer mesh using hexahedral cells created a higher velocity reduction in the wake of the frame than the boundary layer mesh using wedge cells. Looking deeper into the data it was seen that the simulations using the wedge cells in the boundary layer mesh had a less stable wake behind the frame than the ones using the hexahedral cells in the boundary layer mesh. The measurement indicated that the wake behind the frame around the net panel should be wider, meaning more diffusion was needed in the solution. Using the tetrahedral mesh added numerical diffusion, but using a mesh with higher numerical diffusion was not the correct way to add diffusion to the problem, since this was modeling a physical diffusion by a numerical diffusion that could have a different nature. It should be noted that the velocity reduction at 44cm from the centerline for 60° angle of attack in Figure 3-8 was close to the center of the wake behind the frame and was the best measurement for the maximum reduction.
in the wake of the frame. This means that the wake from the frame for case 71 was closer in width to the measured wake, but undershot the maximum reduction, while case 76 had a narrower wake and slightly overshot the maximum reduction. Combining this information and the limited time available for running simulations, the mesh used in case 76 in Table 3.8 seemed the best suited for the problem. There might have been other processes involved with the measurement that added diffusion that were not included in the model. Some of these might have been, vibration of the frame, vortex shedding (that could not be resolved with the coarse mesh used and/or while using a steady state solution) and residual turbulence in the tank water due to previous runs.

3.2.4 Turbulence at the Inlet Boundary

Turbulence in the incoming water could influenced the flow further downcurrent. The turbulence present in the water during the measurements in the tow tank was limited to residual turbulence from the previous run. Since a rather long waiting time between subsequent tank runs was used (>10min), the residual turbulence should have been quite limited. The turbulent kinetic energy \( k \) is defined as

\[
k = \frac{3}{2} u'_{\text{rms}}^2
\]

where \( u'_{\text{rms}} \) is the root mean square of the turbulent velocity fluctuations. A high estimate of the residual turbulence was \( u'_{\text{rms}} = 0.005 \text{m s}^{-1} \) resulting in \( k = 3.75 \times 10^{-5} \text{m}^2 \text{s}^{-2} \).

To test whether the solution was dependent on the incoming turbulence, several cases with different incoming turbulence were setup and run using the case 76 mesh in Table 3.8. To get a feeling for what the different \( k \) and \( \epsilon \) values represent, turbulence intensity, \( I \), turbulence length scale, \( l \), and turbulent viscosity ratio, \( \frac{\mu_t}{\mu} \),
Table 3.9: Different turbulence cases tested modeling the full domain using a 50mm thick porous media.

<table>
<thead>
<tr>
<th>Case</th>
<th>$k$ (m$^2$s$^{-2}$)</th>
<th>$\epsilon$ (m$^2$s$^{-3}$)</th>
<th>$I$ (%)</th>
<th>$I$ (m)</th>
<th>$\mu$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turb1</td>
<td>$2.35\cdot10^{-6}$</td>
<td>$4.9\cdot10^{-8}$</td>
<td>1</td>
<td>0.012</td>
<td>10.1</td>
</tr>
<tr>
<td>Turb5</td>
<td>$1.5\cdot10^{-4}$</td>
<td>$1.25\cdot10^{-6}$</td>
<td>8</td>
<td>0.24</td>
<td>1620</td>
</tr>
<tr>
<td>Turb6</td>
<td>$1.5\cdot10^{-4}$</td>
<td>$2.025\cdot10^{-5}$</td>
<td>8</td>
<td>0.015</td>
<td>100</td>
</tr>
<tr>
<td>Turb9</td>
<td>$3.75\cdot10^{-5}$</td>
<td>$2.5\cdot10^{-7}$</td>
<td>4</td>
<td>0.15</td>
<td>506</td>
</tr>
<tr>
<td>Turb10</td>
<td>$9.38\cdot10^{-6}$</td>
<td>$3.1\cdot10^{-8}$</td>
<td>2</td>
<td>0.15</td>
<td>255</td>
</tr>
</tbody>
</table>

were calculated as well using Equations 2.42 to 2.46. The model runs had incident velocity $u_0 = 0.125$ m s$^{-1}$, since for the same $k$ and $\epsilon$, the effect on the solution was largest at the lowest speed. Incoming $k$ and $\epsilon$ were chosen such that a rather broad set of combinations were represented (see Table 3.9). The main result was to determine whether changing the incoming turbulence would change the structure of the wake behind the net panel enough to change the velocity reduction 2.5m behind the centerline of the net panel. The velocity reduction across the wake 2.5m behind the net panel for all the different cases in Table 3.9 is shown in Figure 3-10. The only observed effect was the added diffusion of the wake from the frame around the net panel. This might be of importance when comparing the velocity reduction behind the net panel at small angles of attack to measured values, since the wake from the frame might dominate those measurements.

As can be seen in Figure 3-10, most of the tested turbulence cases underpredicted the diffusion in the wake of the frame except for turb 5, which overpredicted the diffusion. In turb 5 an unrealistically high $k$ combined with a low $\epsilon$ was used, which produced a very high viscosity ratio. This was not found realistic and the case with the second largest diffusion, turb 9, was chosen for further work.

3.2.5 Conclusions from the Preliminary CFD Runs

The thickness of the porous media did not seem to affect the simulation result significantly at $\alpha = 90^\circ$ and $\alpha = 30^\circ$. Also, using a quite coarse mesh away from
Figure 3-10: Plot of velocity reduction across the wake 2.5m behind the net panel for different turbulence quantities at the inlet boundary. All simulations were run using 2nd order discretization and the Realizable k-ε turbulence model. Incoming velocity was $u_0 = 0.125 \text{ m/s}$ for all cases. The measurement data shown is for $u_0 = 0.5 \text{ m/s}$.

The edges of the porous media did not have a large effect. This led to the conclusion that for the validation of the CFD approach, the net panel should be modeled using a 5cm thick porous media, using a tetrahedral mesh where the edges have the same cell size as was used for the frame around the net, but the cell size should be increasing towards the center of the net panel, with a maximum cell size the same as the thickness of the porous media.

Using the symmetry boundary condition seemed to induce a distortion of the wake behind the frame such that the velocity reduction behind the frame was considerably lower close to the symmetry plane. Therefore, using symmetry and only modeling half the domain was not used in the further work.

There did not seem to be a large difference in using the laminar flow option inside the porous media. This option was not used in further work either.

If the wake behind the frame was to be resolved, the mesh size behind the frame
pipes needed to be small (< 25mm). This was achieved without producing a too large number of cells by using the Hex Core meshing scheme.

3.3 Porous Media Resistance Coefficients

To be able to effectively apply the proposed CFD approach without a lot of iterative simulations to find the correct porous media resistance coefficients, a method needs to be developed to find the coefficients. A method for finding a reasonable choice of porous media resistance coefficients is developed in the following sections, fitting a simplified formulation of the equations solved in the CFD approach to the drag- and lift force measurements on the net panel.

3.3.1 Analytical Approach for Calculating Forces on the Net Panel

Optimal porous resistance coefficients ($D_n$, $D_t$ and $C_n$, $C_t$) were obtained by minimizing the error between porous resistance model predictions and net panel drag and lift measurements. Because it would have been very time-consuming to use the full CFD computer model to generate multiple predictions, a simplified analytical model was used. In the analytical model drag and lift were linearly related to the porous resistance coefficients, so standard analytical error minimization techniques could be employed straightforwardly.

Remembering (Equation 2.36, Section 2.1.5) that pressure drop through porous media is described by

$$\frac{\partial p}{\partial x_i} = S_i = -D_{ij}\mu u_j - \frac{1}{2}C_{ij}\rho u_{mag} u_j. \quad (3.8)$$

If this is in a coordinate system aligned with the principal axes of the net panel ($x_1$
normal to the net panel and $x_2$ and $x_3$ normal to each other, but parallel to the plane of the net panel) the $D_{ij}$ and $C_{ij}$ matrices are of the following form:

$$D_{ij} = \begin{bmatrix} D_n & 0 & 0 \\ 0 & D_l & 0 \\ 0 & 0 & D_t \end{bmatrix} \quad (3.9)$$

$$C_{ij} = \begin{bmatrix} C_n & 0 & 0 \\ 0 & C_l & 0 \\ 0 & 0 & C_t \end{bmatrix} \quad (3.10)$$

Defining the $x_i'$ system as the global coordinate system rotated around the vertical $x_3$-axis, Equation 3.8 can be expressed as

$$\frac{\partial p}{\partial x_i'} = S_i' = -D_i'\mu u_j' - \frac{1}{2} C_i'\rho u_{mag} u_j'$$ \quad (3.11)

where

$$D_i' = R_{ip} R_{jq} D_{pq} \quad (3.12)$$

and

$$C_i' = R_{ip} R_{jq} C_{pq} \quad (3.13)$$

where

$$R = \begin{bmatrix} \cos \alpha & -\sin \alpha & 0 \\ \sin \alpha & \cos \alpha & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad (3.14)$$

Assuming that

$$\frac{\partial p}{\partial x_i'} \sim \frac{(\Delta p)_i}{\Delta x_i'} \quad (3.15)$$

where $\frac{(\Delta p)_i}{\Delta x_i'} (i \text{ not summed})$ is the pressure gradient across the thickness of the
porous media $\Delta x'$ in the $x'$ direction.

Pressure gradients through the porous media were assumed constant through the thickness though pressure on each face could vary. Velocity through the porous media was also assumed evenly distributed spatially across the net panel, and using a porous media with a thickness much smaller than the other dimensions, the drag, $D$, and lift force, $L$, were calculated using Equation 2.94 as

$$D = (\Delta p)_i A'_1 = S'_1 \Delta x'_1 A'_1, \quad L = S'_2 \Delta x'_2 A'_2. \quad (3.16)$$

In Equation 3.16, $A'_1$ is the projected area of the net panel in the $x'$ direction,

$$A'_1 = A \cos \alpha, \quad A'_2 = A \sin \alpha; \quad (3.17)$$

$\Delta x'_1$ is the distance through the net panel in the $x'$ direction,

$$\Delta x'_1 = \frac{t}{\cos \alpha}, \quad \Delta x'_2 = \frac{t}{\sin \alpha}; \quad (3.18)$$

t is thickness, and $\alpha$ is the angle of attack.

The drag and lift forces were calculated as

$$D = S'_1 t A, \quad L = S'_2 t A \quad (3.19)$$

and the drag and lift coefficients, $C_d$ and $C_l$, were calculated as

$$C_d = \frac{2D}{\rho A u'_0^2} = \frac{2S'_1 t}{\rho u'_0^2}, \quad C_l = \frac{2S'_2 t}{\rho u'_0^2} \quad (3.20)$$

An approximation for $u'_i$, the velocity through the porous media in the $x'_i$ direc-
tion, was needed. It was assumed that

$$\bar{u} = \begin{pmatrix} u_0(1 - r_n C_d) \\ 0 \\ 0 \end{pmatrix} \quad \text{(3.21)}$$

where $C_d$ was found from measurements and $r_n$ was an estimate based on CFD simulations of flow through the porous media for a net panel at chosen angles of attack and different speeds. The expression for the velocity reduction is based on the formulation by Leland (1991) in Equation 1.6. It was assumed that the across flow effect from the lift force was negligible. The parameter $r_n$ was calculated from CFD results as

$$r_n = \frac{u_0 - \langle u_{nx} \rangle}{C_d u_0} \quad \text{(3.22)}$$

where $u_0$ was the incoming velocity; $\langle u_{nx} \rangle$ was the average velocity in the tow direction inside the porous media, and $C_d$ was the drag coefficient of the porous media. The parameter $r_n$ changed quite a lot with angle of attack, but did not seem to depend much on speed, and the same value was used for all speeds. The parameter $r_n$ did not seem to be highly affected by the porous media resistance coefficients used, although the range of coefficients tested was limited since all were aimed at representing the net panel tested in Section 3.1. Table 3.10 provides calculated values of $r_n$. In Figure 3-11 a curve is shown that can be used to find values of $r_n$ at other angles of attack than the ones in Table 3.10. The numbers in Table 3.10 are based on CFD simulations performed at $u_0 = 0.5 \text{cm s}^{-1}$ and $u_0 = 0.125 \text{cm s}^{-1}$ using porous coefficients $D_n = 80760 \text{m}^{-2}$, $D_t = 56230 \text{m}^{-2}$, $C_n = 5.356 \text{m}^{-1}$ and $C_t = 1.616 \text{m}^{-1}$. The present approach is not valid for small $\alpha$ since the requirements for Equation 2.94 are not met. The way Equations 3.20
Table 3.10: Values of $r_n$ found from running CFD simulations at different angles of attack

<table>
<thead>
<tr>
<th>$\alpha$</th>
<th>0°</th>
<th>15°</th>
<th>30°</th>
<th>45°</th>
<th>60°</th>
<th>90°</th>
</tr>
</thead>
<tbody>
<tr>
<td>$r_n$</td>
<td>22.8</td>
<td>1.85</td>
<td>0.704</td>
<td>0.395</td>
<td>0.251</td>
<td>0.176</td>
</tr>
</tbody>
</table>

Figure 3-11: The variation of $r_n$ with angle of attack. The figure is based on data with $D_n = 80760m^{-2}$, $D_t = 56230m^{-2}$, $C_n = 5.356m^{-1}$ and $C_t = 1.616m^{-1}$ at two speeds, $u_0 = 0.125m/s$ and $u_0 = 0.5m/s$.

and 3.22 are defined, with $r_n$ coupled to $C_d$ and $\langle u_{nx} \rangle$, seems to make the approach applicable to all $\alpha$, but it should be used with caution for small $\alpha$.

### 3.3.2 Finding the best Porous Media Resistance Coefficients

Three different error functions were minimized in order to identify the best way to find the porous media resistance coefficients $D_n$, $D_t$, $C_n$ and $C_t$. The error functions used were a least squared normalized error (LSNE)

$$LSNE = \frac{1}{N} \sum_{i=1}^{N} \left( \frac{D - D_{\text{measured}}}{D} \right)^2 + \frac{1}{M} \sum_{j=1}^{M} \left( \frac{L - L_{\text{measured}}}{L} \right)^2. \quad (3.23)$$
Table 3.11: Porous coefficients from the different error functions.

<table>
<thead>
<tr>
<th>Error function</th>
<th>$D_n$ (m$^{-2}$)</th>
<th>$D_t$ (m$^{-2}$)</th>
<th>$C_n$ (m$^{-1}$)</th>
<th>$C_l$ (m$^{-1}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$LSNE$</td>
<td>75854</td>
<td>35409</td>
<td>4.8419</td>
<td>1.4439</td>
</tr>
<tr>
<td>$LAE$</td>
<td>76486</td>
<td>84741</td>
<td>5.0727</td>
<td>1.1300</td>
</tr>
<tr>
<td>$LANE$</td>
<td>51730</td>
<td>26379</td>
<td>5.0980</td>
<td>1.6984</td>
</tr>
</tbody>
</table>

A least absolute error (LAE) similar to that used by Zhan et al. (2006)

$$LAE = \frac{\sum |D - D_{measured}| + \sum |L - L_{measured}|}{\sum |D| + \sum |L|}, \quad (3.24)$$

and a least absolute normalized error (LANE)

$$LANE = \frac{1}{N} \sum_{i=1}^{N} \left| \frac{D_i - D_{measured}}{D_i} \right| + \frac{1}{M} \sum_{j=1}^{M} \left| \frac{L_j - L_{measured}}{L_j} \right|, \quad (3.25)$$

A script was written in Matlab to solve the problem of minimizing the error functions for the variables $D_n$, $D_t$, $C_n$ and $C_l$ utilizing a minimizing function, Equation 3.19 and the above mentioned assumptions. The data used for the fitting was tow tank data from Patursson (2007), explained in Section 3.1, where drag, $D$, and lift force, $L$, and velocity reduction behind the net panel, $U_R$, were measured as a function of angle of attack, $\alpha$, (0, 15, 30, 45, 60, 75 and 90°) and tow speed, $u_0$, (12.5, 25, 50 and 75 cm s$^{-1}$). Because the $D$ and $L$ were measured at velocities close to but not exactly equal to the standard velocities (12.5 cm s$^{-1}$, 25 cm s$^{-1}$, 50 cm s$^{-1}$ and 75 cm s$^{-1}$); $C_d$ and $C_l$ were first calculated and $D$ and $L$ were then reproduced using the standard velocities. A least squares fitting function built-in to Matlab was used to solve for $LSNE$. A function was written to solve for the absolute error functions. Plots representing the results from the fits are shown in Figures 3-12 to 3-14, and the associated porous media resistance coefficients are given in Table 3.11.

The fitted velocity reduction that is compared to the measured velocity reduc-
Figure 3-12: Data from the fit using $LSNE$. The lines correspond to the optimized analytical model; the crosses (×) to measurements.
Figure 3-13: Data from the fit using LAE. The lines correspond to the optimized analytical model; the crosses (x) to measurements.
Figure 3-14: Data from the fit using LANE. The lines correspond to the optimized analytical model; the crosses (×) to measurements.
Figure 3-15: The variation of $r$ with angle of attack. The figure is based on data with $D_n = 80760 \text{m}^{-2}$, $D_t = 56230 \text{m}^{-2}$, $C_n = 5.356 \text{m}^{-1}$ and $C_t = 1.616 \text{m}^{-1}$ at two speeds, $u_0 = 0.125 \text{m s}^{-1}$ and $u_0 = 0.5 \text{m s}^{-1}$.

The variation in Figures 3-12 to 3-14 was calculated as

$$U_r = rC_d$$  \hspace{1cm} (3.26)

where $r$ was found in a similar manner as $r_n$ and was calculated as

$$r = \frac{u_0 - u_{2.5}}{C_d u_0}$$  \hspace{1cm} (3.27)

where $u_{2.5}$ is the current velocity 2.5m behind the center of the net panel found from CFD simulations. Figure 3-15 shows the $r$-values used.

The coefficients found using the LAE error function seemed to overpredict the velocity dependence of the drag coefficient and, hence, also of the velocity reduction, and the lift coefficient seemed to have a reversed velocity dependence. The coefficients found using the LSNE and the LANE error functions showed quite similar results, but since the LSNE error function depended on the squared error, this error function had a higher response to bad data points. This was the reason for the large velocity dependence of the lift coefficient observed in Figure 3-12. If
Table 3.12: Porous coefficients for the different data sets from Rudi et al. (1988)

<table>
<thead>
<tr>
<th>S</th>
<th>(D_n) ((\text{m}^{-2}))</th>
<th>(D_l) ((\text{m}^{-2}))</th>
<th>(C_n) ((\text{m}^{-1}))</th>
<th>(C_l) ((\text{m}^{-1}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.130</td>
<td>112250</td>
<td>34352</td>
<td>3.3015</td>
<td>1.9928</td>
</tr>
<tr>
<td>0.243</td>
<td>152760</td>
<td>37283</td>
<td>6.1493</td>
<td>2.8146</td>
</tr>
<tr>
<td>0.317</td>
<td>845370</td>
<td>105900</td>
<td>12.401</td>
<td>6.5737</td>
</tr>
</tbody>
</table>

the lift force measured at 12.5cm s\(^{-1}\) and 60° and at 75cm s\(^{-1}\) and 45° were omitted, the fit using \(LSNE\) and \(LANE\) would be almost the same. Therefore, it was decided to use the coefficients from the \(LANE\) error function for further analysis.

To test the robustness of the routine for finding porous coefficients, it was tested on data measured by Rudi et al. (1988). The reported data included drag and lift coefficients for several net panels at different angles of attack. The measured and calculated drag and lift coefficients for three net panels of different solidities are shown in Figure 3-16, and the associated porous coefficients are given in Table 3.12. It seemed like the drag at \(\alpha = 0°\) was overpredicted, at least for some of the measurements, so it was omitted.

From these tests of the method where the \(LANE\) error function has been used to find porous coefficients for four different nets of different solidities, it seems like the method is quite robust and generally fits the data sets well. It is important to note that at the present time the values for \(r_n\) and \(r\) have only been validated against data for one net of moderately low solidity. In the here mentioned processes of finding porous coefficients the model has been fitted against both drag and lift data to get the best overall fit. Zhan et al. (2006) fit their model only against drag data, which might give a better fit in this case as well. The question is then how important the lift force is when it comes to modeling fish cages? This will not be addressed at this point, but could be a topic for further research.
Figure 3-16: Data from the fit of the data from Rudi et al. (1988) using LANE. (a) $S = 0.13$, (b) $S = 0.243$ and (c) $S = 0.317$. The lines refer to the optimized analytical model; the symbols, to measurements.
Table 3.13: Angles of attack and speeds used in the simulation runs that are compared to measured data.

<table>
<thead>
<tr>
<th>Speed (m.s(^{-1}))</th>
<th>Angle of attack (°)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.125</td>
<td>0 15 30 45 60 90</td>
</tr>
<tr>
<td>0.5</td>
<td>0 15 30 45 60 90</td>
</tr>
</tbody>
</table>

3.4 Results from Computational Fluid Dynamics

CFD simulations were run modeling the net panel and frame at a few different angles of attack and two different speeds such that the CFD simulation results could be compared to the tow tank measurements. It was chosen not to run simulations corresponding to all data points in the measurement series; only 6 angles of attack and two representative speeds, 12.5cm.s\(^{-1}\) and 50cm.s\(^{-1}\) (Table 3.13), were used due to the long computation times needed to run the simulations for all speeds.

The mesh used for the simulations was the same as for Case 76 in Table 3.8. The standard wall functions were used. All walls were moving walls except for the surface that was modeled as a wall with zero shear force. The turbulence model used was the realizable \(k - \epsilon\) model, and the incoming turbulence was set to be the quantities used for Case 9 in Table 3.9. The porous coefficients used were the ones found using the LANE error function in Table 3.11.

The variables \(C_d\), \(C_l\) and \(U_r\) were compared as shown in Figure 3-17, and the difference between measured and modeled data was generally small with a few exceptions.

- The difference between measured and modeled \(U_r\) was quite large at small angles of attack. This was mainly due to the fact that the wake from the frame around the net panel was narrower in the model than what was measured, and the high \(U_r\) at \(\alpha = 15°\) in the measured data was due to the wake from the frame, while the effect from the frame was smaller or nonexistent in the CFD prediction. There was larger diffusion of the wake behind the model at
Figure 3-17: Comparing the results from the CFD simulations to measurement data.
slower speeds, and this was also reflected in the comparison with a smaller
difference between measured and modeled data at $u_0 = 0.125 \text{m s}^{-1}$.

- The measured data point of $C_l$ at $u_0 = 0.125 \text{m s}^{-1}$ and $\alpha = 60^\circ$ looks suspici­
cious and might be a bad data point.

- The measured $C_l$ was less than zero at $\alpha = 90^\circ$. This might have been a
correct measurement due to some anti-symmetries in the net panel, but the
model did not have the capability to predict a non-zero $C_l$ at $\alpha = 90^\circ$.

### 3.4.1 Accuracy of the Porous Resistance Coefficient Procedure

To test whether the procedure for finding the porous coefficients was giving accurate
enough results, the effect of changing the porous resistance coefficients some value
around the value found using the method in Section 3.3.2 was explored. This
was done by calculating the same error function as used for the fit in Section
3.3.2, $LANE$, between the CFD results and the measured data. It was too time
consuming to run the CFD simulations for all the data points in the measurement
series, and a few data points were chosen that showed a similar error function as the
whole data set. The choice of data points was examined using the analytical model
(Figure 3-18) where the porous resistance coefficients were offset a percentage of
the fitted coefficients, found using the $LANE$ error function, given in Table 3.11.
The offset was done for one coefficient at a time while the other three coefficients
were held fixed at the fitted value. Figure 3-18 shows that the analytical error
function using only four data points, $u_0 = 0.125 \text{m s}^{-1}$ and $u_0 = 0.5 \text{m s}^{-1}$ at $\alpha = 45^\circ$
and $\alpha = 90^\circ$, had similar behavior to the error function using all the data points
in the measurement series. The error itself was smaller due to the fact that these
measurements generally had smaller errors than many of the other data points.
Figure 3-18: Sensitivity of analytical method to offset in porous resistance coefficients. Each line represents the variation of the error function, \( LANE \), as a function of the offset of the respective coefficient, while the other coefficients are not changed. The figure on the left was made using all data points in the measurements, while the figure on the right only included four data points, \( u_0 = 0.125 \text{m/s} \) and \( u_0 = 0.5 \text{m/s} \) at \( \alpha = 45^\circ \) and \( \alpha = 90^\circ \).

The accuracy of the porous resistance coefficient procedure was tested by running the model with the porous resistance coefficients varied around the predicted value and calculating the error function similar to what was done for Figure 3-18, but for the limited number of data points. The model was run for the four scenarios \( u_0 = 0.125 \text{m/s} \) and \( u_0 = 0.5 \text{m/s} \) at \( \alpha = 45^\circ \) and \( \alpha = 90^\circ \) for each of the sets of porous resistance coefficients. For each of the sets of porous resistance coefficients, the \( LANE \) error function was calculated in addition to another error function, \( (LANE_a) \) where the velocity reduction was included

\[
LANE_a = \frac{1}{K} \sum_{i=1}^{K} \left| \frac{D - D_{measured}}{D} \right| + \frac{1}{M} \sum_{j=1}^{M} \left| \frac{L - L_{measured}}{L} \right| + \frac{1}{N} \sum_{n=1}^{N} \left| \frac{U_r - U_{r,measured}}{U_r} \right| \tag{3.28}
\]

The results are shown in Figure 3-19.
Figure 3-19: Sensitivity of CFD method to offset in porous resistance coefficients. Each line represents the variation of the error function, $LANE$, as a function of the offset of the respective coefficient while the other coefficients were not changed. The figures were made using the four data points, $u_0 = 0.125\text{m s}^{-1}$ and $u_0 = 0.5\text{m s}^{-1}$ at $\alpha = 45^\circ$ and $\alpha = 90^\circ$. The figure on the left shows the original $LANE$ error function (Equation 3.25), and the figure on the right plots the $LANE_u$ error function where the current reduction was included (Equation 3.28).

The results show that the coefficients with the largest effect on the error were chosen well. $C_n$ which had the largest effect on the calculated error was chosen within a few percent, while $C_t$ which also had a fairly large effect on the error was underpredicted by around 10%. The other two coefficients were not predicted very accurately, but they had a very limited effect on the error, especially $D_t$.

3.5 Discussion

The method of modeling the flow around a net panel gave predictions which seemed to agree well with measured data for both drag and lift force on the net panel and current reduction behind the net panel. Also the proposed analytical method of finding the optimal porous resistance coefficients seemed to give coefficients for which model results agreed fairly well with the data. The proposed mesh and
setup of the CFD model gave good results overall, but the wake of the frame around the net panel was not modeled well, and this resulted in a rather large discrepancy between modeled and measured current reduction behind the net panel at small angles of attack. Since the measured current reduction data was not used in the method for finding the porous resistance coefficients, the discrepancy between measured and modeled velocity reduction at small angle of attack did not affect the coefficients found, but the comparison between measured and modeled velocity reduction served as a first validation of the approach, which was promising since the difference was very small except for the mentioned discrepancy at small angle of attack.
CHAPTER 4

GRAVITY CAGE STUDIES

Gravity cages are mainly built using a floating collar supporting a net pen hanging from the collar with some weight at the bottom perimeter of the net to reduce deformation due to current. The most used geometry for gravity cages in exposed areas are circular cages made from high density polyethylene (HDPE) pipes with a cylindrical net and often a circular weight ring is attached to the bottom of the net. An illustration of a gravity cage is shown in Figure 4-1.

In this part of the study the model was validated by comparing predictions with tow tank measurements for a small gravity-type cage performed at the U. S. Naval Academy (USNA). The model was then applied to a clean and a biofouled commercial size cage and compared to measurements of current reduction inside and behind a similar cage by Patursson and Simonsen (2008).

4.1 Tow Tank Experiments

The small gravity type cage for tow testing at the USNA is shown in Figure 4-2 and approximates a standard gravity cage, but reduced in size. The cage was made with two stiff octagonal rims, one on the top and one on the bottom for weight. The enclosure was formed using a bottom net within the lower rim and sides between the rims. The width of the cage, measured between midpoints of opposite rim sections, was 3.124m, and the height was 1.683m. The octagonal shape is not the most used for gravity cages, but it is close to the circular shape mostly used,
and the Bridgestone cages (http://akva-trade.com/) do actually have this shape.
The octagonal shape can be made in sections and assembled on site, which makes
modeling, manufacturing and transport easier than for a circular shape.

The experimental program included measurement of drag on the cage at different
speeds when towed from the top rim only and from both rims. Current reduction
was measured at different positions inside the cage and in the wake region. Net
deformation was recorded when towing from the top rim only.

Not all of the test results were used for the model evaluation, so only drag force
and current reduction for the double tow arrangement are presented in this disser­
tation (Section 4.1.6). The rest of the results have been summarized by Fredriksson
et al. (2007).

4.1.1 Fish Cage Particulars

The small fish cage was constructed using nominal 2in diameter (60mm OD) sched­
ule 40, steel pipe and knotless nylon net. The steel pipe was used for the upper and
lower rim assemblies in an octagon shape (Figure 4-2) creating an in-tow length
of 3.124m (123in). The top rim was placed above the free surface during the tow
tests supported by four steel cables hung vertically from the tow carriage (Figure 4-3). The bottom rim was built to be geometrically identical, but was filled with lead shot so the entire assembly weighed 1957N (440lbf). It should be noted that the buoyancy force on the bottom rim was 275N. The distance between the two rim assemblies was 1.683m (66.25in). Knotless net was strung between the two rim assemblies in a square mesh configuration. The net was the same net as was used for the net panel tests in Section 3.1 \(d = 2.8\text{mm} \text{ and } \lambda = 29\text{mm}\).

The solidity ratio was found using the same method as for the net panel test where the pixels of the background color were sorted from the pixels of the net color in a digital picture of the net. The solidity \(S\) was estimated to be 0.20 when the net was stretched on the frame, but when the net was attached to the cage as seen in Figure 4-2, it was only stretched in the vertical direction, and the solidity increased to \(S = 0.22\).
4.1.2 Approach

The fish cage was used in a series of tow tests to measure the drag, flow reduction and cage bottom deflection characteristics. Two tow configurations were employed (Figure 4-3). In one set of tests, the cage was towed with two cables. The cables were connected horizontally between the upper rim and a short tow post as well as the lower rim and a long tow post. During this set of tests, tow cable tension was measured in both tow cables, and flow reduction characteristics through the netting were measured at 12 locations. Only one tow cable attachment, to the upper rim of the cage, was used during the second set of tests. In this portion of the experimental program, cable tension and lower rim displacement was observed as well as flow reduction at 5 locations. Both sets of tests were conducted at speeds of 0.125, 0.25, 0.50, 0.75 and 1.00m s\(^{-1}\). To ensure that the water in the tank had settled sufficiently between runs, a waiting time of at least 10 minutes was used. In addition, current meter output was observed as well. If there was observed water movement after the 10 minutes \((u > 1\text{cm s}^{-1})\), further waiting time was used.
4.1.3 Equipment

Testing Facility

The tests were performed in a 116 m x 7.9 m x 4.9 m tank facility at the Hydromechanics Laboratory at the USNA. The towing capabilities included two tow carriage assemblies as shown in the Figure 4-4 schematic. During the tests, tow carriage speed, fish cage drag, fluid velocities at multiple locations and deflection of the bottom rim were measured. The testing equipment at the USNA included two force gages, two Nobska MAVS-3 (www.nobska.net) current meters and an underwater camera. The following sections describe the equipment used during the tests except for the camera measuring deflection of the bottom rim, since these measurements are not used in the present study.
Tow Carriages

Tow carriage speed was measured using a wheel and tachometer system mounted along the carriage rail. The tachometer was a BEI (www.beiied.com) rotation encoder with a resolution of 1200 pulses per revolution. The encoder was driven by a 114 mm (4.5 inch) diameter steel wheel that rides on one of the round steel carriage rails supporting the carriage. Voltage output from the converter was calibrated in terms of carriage speed at least twice per year using a distance over time approach.

Force Gages

Drag force was measured using two modular force gages. The gages consisted of a 102 mm (4 in) block made of ARMCO 17-4PH stainless steel with flexures sensitive to forces along one axis. The benefit of using this type of gage was that once aligned in the direction of the intended force measurement, the instrument was nearly invulnerable to forces or moments from unexpected sources. When the gages were loaded along their sensitive axis, two opposite surfaces of the block moved relative to each other. This motion was sensed by a waterproof, variable-reluctance displacement transducer. The manufacturer's specification for linearity was +/- 0.25 percent of full load though calibrations showed that the response to be even more linear. The force gages were calibrated at the Hydromechanics Laboratory at the USNA.

Current Meters

Fluid velocities were measured with two Nobska MAVS current meters. The MAVS current meter used differential travel time measurements to estimate velocity. The advantage of using the MAVS current meter in a clean tank facility was that, unlike acoustic doppler instruments, the water did not need to be seeded with particulates to make the measurements. The MAVS also recorded temperature, tilt
Figure 4-5: The MAVS current meters were calibrated in a 33 meter long tow tank facility before and after the fish cage tow tests.

and orientation, and provided velocity in earth or fixed coordinates. The MAVS current meters were calibrated before and after the cage tow tests using a different 33 meter tow tank, also within the Hydromechanics Laboratory (Figure 4-5). The instruments were "towed" while attached to the carriage at speeds of 0.12, 0.25, 0.50, 0.75 and 1.00 m/s, each with two replicates. Two zero offsets were also measured.

The current meters were configured to measure u (local x-axis), v (local y-axis) and w (local z-axis) components of the flow. The x-axis was orientated along the centerline of the tank. The y-axis was horizontal and orientated perpendicular to the tank centerline. The z-axis was orientated vertically. Since the x-axis of the current meters did not line up very well with the centerline of the tank, the u and v components had to be calibrated separately, and then the speed could be calculated, which was used to compare to the incoming flow speed (carriage speed). The current meters were calibrated using the same orientation as for the measurements, and calibration curves were generated for the u and v components separately. To make the calibration curves, the local components of the carriage
speed had to be found first, and these were calculated using the average angle between the local y-axis and the travel direction of the carriage calculated from the measured \( u \) and \( v \) values. The data obtained during calibration was very linear, but showed some scatter. The observed difference between calibrated current speed and carriage speed was less than 1.5% of the measured speed. During data processing, the calibrated \( u \) and \( v \) components of the flow were found, and the speed was calculated.

### 4.1.4 Experimental Setup

**Tow Tank**

The towing tests were conducted with the forward and aft towing carriages connected (Figure 4-4). During the tow experiments (see Figures 4-3, 4-6 and 4-7), the fish cage was suspended from the aft tow carriage using steel cables with the
upper rim placed approximately 45mm above the free surface. The cage was towed in two different configurations with separate towing post assemblies located on the forward towing carriage. The first set utilized two force gages attached to separate tow posts. The longer tow post had a length of 2.44m and a width of 28cm that tapered to 7.6cm. The long tow post was faired and made of stainless steel. The shorter tow post had a length of 0.508m and was terminated just above the free surface. Each tow post had a force gage mounted to the lower end. The two tow post assemblies are shown on Figure 4-6 connected to the forward tow carriage. The second set of tow tests utilized only one force gage connected to the shorter tow post. The cage is shown suspended from the aft tow carriage in Figure 4-7.

After the tows pulling from both rims were finished, and the single tow post measurements were started, it was realized that the tow cable from the shorter tow post to the upper rim was not tightened up properly, which lead to a setback of the cage during the tow. This setback resulted in inclination of the suspension cables so that they took up some of the drag force, which caused the measured drag force to be less than the real drag force on the cage. When this was discovered, the shorter tow post was moved further away from the cage, and the setback of the cage during tows was reduced to almost zero.

4.1.5 Data Processing

Force Gage Measurements

Force gage measurements were obtained from both the double and single tow configurations. A time series average over a representative section of the time series was used to calculate the drag force, similar to what is seen for the net panel in Figure 3-6. A new zero reading for each of the runs was included in the calculation of the force. Where several runs were made for the same measurement, the average
Current Velocity Measurements

Current measurements were primarily collected during the double tow configuration tests, though a limited number were also acquired during the single tow tests. The post calibration information was applied to the data sets downloaded from the instruments. Each measurement represents a time series average of a representative section of the time series, similar to what is seen for the net panel in Figure 3-6, referenced to a zero reading made immediately prior to the start of the run. For each measurement, the measured speed was normalized with the average tow speed for the corresponding run and then, if several runs were made for the same measurement, the average value was used.
4.1.6 Results

Fish Cage Drag

The results are given in Table 4.1 for drag force as a function of tow speed for the different tow configurations and standard deviation of the replicated measurements. The time and replicate-averaged data results for the tow speed and force block measurements are provided in Table 4.1, where the two first columns of data (Top Only\textsuperscript{1} and Top Only\textsuperscript{2}) show the difference before and after the upper tow post was pulled further away from the cage, such that the tow cable was better tightened up. Columns four and six (Top\textsuperscript{1} and Bottom\textsuperscript{1}) are the measurements on the top and bottom rims, respectively, during the double tow point measurements. The final column is the total drag force measured during the double tow point configuration calculated as the sum of the two previous columns (Top\textsuperscript{1} and Bottom\textsuperscript{1}).

It can be seen in Table 4.1 that the adjustment of the tow post had a significant impact on the drag force measurement on the upper rim. When the tow cable was slack (before adjustment), the drag force increased the cage set back, and some of the drag force was taken up by the non-vertical suspension cables. It can be assumed that the measurement on the lower rim was unaffected by this movement, while the measurement on the upper rim was more affected, especially those performed at slow speeds.

Since the values used in the present study (the measurements using two tow points) were obtained before the adjustment of the tow post, there was an error associated with the measurement on the upper rim. There was almost no drag force measured on the upper rim at $u_0 = 0.126 \text{ m s}^{-1}$ and $u_0 = 0.25 \text{ m s}^{-1}$. This was clearly not true, and the drag force on the combined tow from both the top and bottom rim was underestimated. This needed to be taken into account when data from the CFD simulations was compared to the measurements.
Table 4.1: Drag force measurements (F) and standard deviation (SD) with the single (first three columns) and double (last five columns) cable attachments. All data but the first column (Top Only\(^1\) (N)) include replicates. There are three replicates of the tows on the upper rim and six replicas of the measurements on the double rim tows.

<table>
<thead>
<tr>
<th>Speed (m/s)</th>
<th>Top Only(^1) F (N)</th>
<th>SD (N)</th>
<th>Top Only(^2) F (N)</th>
<th>SD (N)</th>
<th>Top(^1) F (N)</th>
<th>SD (N)</th>
<th>Bottom(^1) F (N)</th>
<th>SD (N)</th>
<th>Top and bottom(^1) F (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.126</td>
<td>2.70</td>
<td>1.01</td>
<td>0.671</td>
<td>0.18</td>
<td>13.4</td>
<td>0.73</td>
<td>14.1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.25</td>
<td>33.7</td>
<td>1.08</td>
<td>3.11</td>
<td>0.29</td>
<td>52.8</td>
<td>1.55</td>
<td>55.9</td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.50</td>
<td>-</td>
<td>7.81</td>
<td>70.3</td>
<td>3.56</td>
<td>214</td>
<td>2.07</td>
<td>284</td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.75</td>
<td>-</td>
<td>722</td>
<td>244</td>
<td>5.33</td>
<td>468</td>
<td>3.52</td>
<td>712</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.00</td>
<td>-</td>
<td>1195</td>
<td>491</td>
<td>4.98</td>
<td>819</td>
<td>7.39</td>
<td>1310</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

\(^1\) Before tow post adjustments
\(^2\) After tow post adjustments

Since there were replicates of the measurements, the standard deviation was calculated to give better insight into the variation between repeated measurements. The standard deviations are shown in Table 4.1. It can be seen from the calculated standard deviations that this kind of error is not as important as the problem with the tow post setting since the standard deviation of replicates is generally less than 2% of the measured force.

**Current Velocities**

The current measurements were processed to obtain flow reduction results by normalizing the measured speeds with the average tow carriage speed. The flow reduction results for the double point tow configuration are provided in Table 4.2 as tabulated values of normalized velocity measurements as a function of carriage speed and current meter position. For two of the measurements four replicates existed (Table 4.2). The data are arranged in three groups each representing a transect in the x-direction (along the tank) with constant y and z coordinates. A transect across the tank (y-direction) can also be generated from the data at x = 4.57m when the measurement at x = 4.34m is included.
Table 4.2: Normalized velocity measurements for the double point tow configuration. The origin was located at the still water level in the center of the cage. The x-axis was directed along the axis of the tank toward the beach in the tank. The z-axis was directed vertical downwards and the y-axis was directed across the tank according to the right hand rule (to the right when looking in the positive x-direction).

<table>
<thead>
<tr>
<th>Position</th>
<th>12.5cm/s</th>
<th>25cm/s</th>
<th>50cm/s</th>
<th>75cm/s</th>
<th>100cm/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>x (m)</td>
<td>y (m)</td>
<td>z (m)</td>
<td>u/u₀</td>
<td>u/u₀</td>
<td>u/u₀</td>
</tr>
<tr>
<td>-0.83</td>
<td>0</td>
<td>0.91</td>
<td>0.895</td>
<td>0.913</td>
<td>0.907</td>
</tr>
<tr>
<td>0.00</td>
<td>0</td>
<td>0.91</td>
<td>0.857</td>
<td>0.857</td>
<td>0.891¹</td>
</tr>
<tr>
<td>0.83</td>
<td>0</td>
<td>0.91</td>
<td>0.882</td>
<td>0.848</td>
<td>0.874</td>
</tr>
<tr>
<td>1.80</td>
<td>0</td>
<td>0.91</td>
<td>0.757</td>
<td>0.776</td>
<td>0.767</td>
</tr>
<tr>
<td>3.05</td>
<td>0</td>
<td>0.91</td>
<td>0.680</td>
<td>0.684</td>
<td>0.723²</td>
</tr>
<tr>
<td>4.57</td>
<td>0</td>
<td>0.91</td>
<td>0.792</td>
<td>0.727</td>
<td>0.713</td>
</tr>
<tr>
<td>0.00</td>
<td>0</td>
<td>1.37</td>
<td>0.812</td>
<td>0.830</td>
<td>0.858</td>
</tr>
<tr>
<td>3.05</td>
<td>0</td>
<td>1.37</td>
<td>0.679</td>
<td>0.682</td>
<td>0.694</td>
</tr>
<tr>
<td>0.00</td>
<td>1.08</td>
<td>0.91</td>
<td>0.949</td>
<td>0.909</td>
<td>0.894</td>
</tr>
<tr>
<td>0.83</td>
<td>1.08</td>
<td>0.91</td>
<td>0.868</td>
<td>0.913</td>
<td>0.869</td>
</tr>
<tr>
<td>4.57</td>
<td>1.08</td>
<td>0.91</td>
<td>0.725</td>
<td>0.770</td>
<td>0.737</td>
</tr>
<tr>
<td>4.34</td>
<td>1.64</td>
<td>0.91</td>
<td>0.846</td>
<td>0.782</td>
<td>0.775</td>
</tr>
</tbody>
</table>

¹ Data set included four replicates. Std Dev = 0.46cm s⁻¹ and Std Dev/u₀ = 0.0092
² Data set included four replicates. Std Dev = 0.51cm s⁻¹ and Std Dev/u₀ = 0.0102

4.1.7 Discussion

One of the primary goals of performing these tests was to have a data set of "full scale" values to examine the validity of numerical model techniques. For the present work the data was used for the validation of the CFD approach explained in Chapter 2 for modeling the flow inside and around a fish farming cage.

It is important to have an understanding of the error associated with the measurements. The force measurements included a fair number of replicates, and the standard deviation of the measurements combined with the error induced by the adjustment of the tow posts indicated that a rather large error was associated with these measurements. The most accurate portion of the drag force data was probably the measurements towing from only the top rim after the tow post adjustment mentioned in Section 4.1.4 (Top Only² in Table 4.1), but for low speeds (<50cm s⁻¹)
where the deformation of the cage is negligible.

The setup for measuring drag force on the cage was not effective in producing accurate results. The error was due to the horizontal component of the force in the support cables generated when the cage was displaced in the horizontal direction. This error was increased due to the use of fairly short support cables and the use of steel cable for the tow line. The weight of the steel cable increased the horizontal movement of the cage, and the short supports had a larger impact on the drag measurement than longer supports would have for the same horizontal displacement. On the positive side, the present approach made it possible to use a quite heavy weight in the bottom rim without the use of a large flotation in the upper rim that would have a large impact on the flow and drag on the cage. If the same approach were to be used again, longer supports should be used, and a light-weight non-stretch tow cable (e.g. spectra twine) should be used.

For the current measurements there were generally no replicates, but the standard deviation for the two available measurements with replicates agrees well with the observed error estimate during calibration of $\pm 1.5\%$ of measured velocity (Section 4.1.3). One of the major error sources was a possible residual current in the tank after the previous run. An estimate of the maximum residual current is $1\text{cm}\text{s}^{-1}$, since this was used as the criteria for starting a new run. This was of much higher importance for the low velocities than for the faster velocities. The total error should then be a combination of these two error sources.

Due to the large error associated with the drag force measurement and the more accurate current measurement, the drag data was considered secondary to the current data in the validation process.
4.2 CFD Application and Validation

As a validation of the CFD model described in Chapters 2 and 3, the model was applied to the cage tow test setup performed at the USNA explained in Section 4.1. The model was tested with regards to changes in the mesh, different turbulence models, turbulence quantities and changes in the porous resistance coefficients. The CFD results were compared to each other and to the data from the tow tests. The net used in the USNA test was the same as was used in the net panel tests at UNH (Section 3.1), but due to the difference in stretching, the solidity of the net in the USNA test was higher than in the UNH test. This required adjustment of the porous resistance coefficients to account for the increased solidity.

4.2.1 New Porous Resistance Coefficients

Due to the higher solidity of the net in the cage than in the net panel tested in Section 3, there was a need to find a new set of porous media resistance coefficients that better describe the resistance of the higher solidity net in the cage. To find new porous coefficients corresponding to the higher solidity of the net in the cage, the porous resistance coefficients from Table 3.11 and 3.12 were plotted as a function of solidity, and curve fits with reasonable fits were produced (Figure 4-8).

These fits were not found good enough for a direct reading of new porous resistance coefficients, but were used to find the increase in the porous resistance coefficients that was then added to the coefficients found from net panel tests in Section 3.3.2. $D_n$ with solidity $S = 0.22$ ($D_n(0.22)$) was calculated as

$$D_n(0.22) = D_n(0.20) + \hat{D}_n(0.22) - \hat{D}_n(0.20) \quad (4.1)$$

where $\hat{D}_n$ was the $D_n$ found from the curve fit. The coefficients found by increasing the solidity of the net from 0.20 to 0.22 are given in Table 4.3. The coefficients for
a 2cm thick porous media are also provided. They were calculated by multiplying the coefficients for the 5cm thick porous media by the ratio between the thicknesses \( \left( \frac{5}{3} \right) \).

### 4.2.2 Validation Results

The domain used for the CFD validation process had the same cross sectional area as the tow tank (width = 7.9m and height = 4.9m) and a length of 50m. The cage was built using the undeformed dimensions of the cage (octagonal shape with length from side to side = 3.124m and depth = 1.683m). The cage was modeled stationary...
Table 4.3: Porous media resistance coefficients from the net panel tests and the porous resistance coefficients found by increasing the solidity of the net from 0.20 to 0.22. The coefficients for the lower thickness (t) were found by multiplying the coefficients for the higher thickness by the ratio between the thicknesses.

<table>
<thead>
<tr>
<th>S</th>
<th>t (mm)</th>
<th>$D_n$ (m$^{-2}$)</th>
<th>$D_t$ (m$^{-2}$)</th>
<th>$C_n$ (m$^{-1}$)</th>
<th>$C_t$ (m$^{-1}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.20</td>
<td>50</td>
<td>51730</td>
<td>26379</td>
<td>5.0980</td>
<td>1.6984</td>
</tr>
<tr>
<td>0.22</td>
<td>50</td>
<td>100100</td>
<td>32360</td>
<td>5.952</td>
<td>2.105</td>
</tr>
<tr>
<td>0.22</td>
<td>20</td>
<td>250250</td>
<td>80900</td>
<td>14.88</td>
<td>5.262</td>
</tr>
</tbody>
</table>

in the tank, and the towing of the cage was modeled by the water flowing from one end of the tank to the other. The inlet velocity used was the same as the tow speed. The surface was modeled as a stationary ceiling (using the wall boundary condition) with no shear force, and the side walls and bottom were modeled as moving walls with the same velocity as the inlet velocity. The cage was positioned from the surface and down, with the center of the cage placed (10m) downcurrent from the in-flow end of the tank, centered on the width of the tank. The net was described using a thin volume of porous media, which was centered on the position of the net, which has a much smaller thickness than the porous media. Two thicknesses were tested - 2cm and 5cm. For most of the meshes only the net was modeled, and the bottom rim and the tow post were omitted from the model except for one mesh that included the bottom rim. The cage geometry using the 5cm thick porous media without the bottom rim can be seen in Figure 4-9.

The mesh outside and inside the cage was generally created using the Hex Core meshing scheme, while the mesh in the porous media describing the net was created using the TGrid scheme. One mesh was created using only the TGrid meshing scheme. Planes were created for controlling the grid size in some meshes. These planes were defined inside all parts of the net that were parallel to the flow, centered on the thickness, and they extended (5m) behind the cage. A mesh size was defined inside the porous media, and these planes were then used to define a smaller mesh size inside the parts of the net that were parallel to the flow and the region behind
Table 4.4: Meshes tested and obtained drag force on the cage. $M$ means mesh size. The porous resistance coefficients used are the ones given in Table 4.3 for solidity $S = 0.22$.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thickness of porous media (mm)</td>
<td>50</td>
<td>50</td>
<td>20</td>
<td>20</td>
<td>50</td>
</tr>
<tr>
<td>Inclusion of bottom ring</td>
<td>No</td>
<td>No</td>
<td>No</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>$M$ inside porous media (net) (mm)</td>
<td>50</td>
<td>50</td>
<td>20</td>
<td>20</td>
<td>50</td>
</tr>
<tr>
<td>$M$ inside nets parallel to flow (mm)</td>
<td>50</td>
<td>20</td>
<td>20</td>
<td>20</td>
<td>20</td>
</tr>
<tr>
<td>$M$ behind nets parallel to flow (mm)</td>
<td>-</td>
<td>20</td>
<td>20</td>
<td>-</td>
<td>20</td>
</tr>
<tr>
<td>Mesh scheme inside porous media</td>
<td>TGrid</td>
<td>TGrid</td>
<td>TGrid</td>
<td>TGrid</td>
<td>TGrid</td>
</tr>
<tr>
<td>Mesh scheme outside porous media</td>
<td>Hex Core</td>
<td>Hex Core</td>
<td>Hex Core</td>
<td>Hex Core</td>
<td>Hex Core</td>
</tr>
<tr>
<td>Growth factor used in size functions</td>
<td>1.2</td>
<td>1.2</td>
<td>1.2</td>
<td>1.2</td>
<td>1.2</td>
</tr>
<tr>
<td>Maximum cell size allowed (m)</td>
<td>0.5</td>
<td>0.5</td>
<td>0.5</td>
<td>0.5</td>
<td>0.5</td>
</tr>
<tr>
<td>Drag @ $u_0 = 0.125\text{m s}^{-1}$ (N)</td>
<td>28.25</td>
<td>29.68</td>
<td>29.71</td>
<td>29.52</td>
<td>30.32</td>
</tr>
<tr>
<td>Drag @ $u_0 = 0.500\text{m s}^{-1}$ (N)</td>
<td>381.6</td>
<td>406.6</td>
<td>405.2</td>
<td>402.0</td>
<td>422.0</td>
</tr>
</tbody>
</table>

those parts of the net. This was generally the region of the domain that included the largest velocity gradients. A size function was used to control the increase in mesh size away from these planes and volumes. The growth rate was set to 1.2 and the maximum size was 0.5m for all meshes. An explanation of all the meshes tested is provided in Table 4.4. Mesh 2 can be seen in Figure 4-10.

The porous media resistance coefficients were defined as explained in Section 4.2.1 and the coefficients for $S = 0.22$ in Table 4.3 were used. All of the meshes were tested using 2nd order discretization, the standard $k - \epsilon$ (SKE) turbulence model and node based derivative calculation. Two velocities were used, 0.125m s$^{-1}$
Figure 4-10: Mesh used for CFD of small cage. The upper two figures are a top view of a horizontal cut through the mesh in the middle of the cage, and a scaled up small section for details. The three lower figures show a vertical cut through the center of the domain. First the full domain is shown, then a shorter section of the domain showing the full width, and last a scaled up small section for details. The octagonal structure on the top figure and rectangular structure on the lower figures is the cage, and the regions with increased mesh density behind the cage can also be seen as tails behind the cage.
Figure 4-11: Velocity distribution around and inside the cage. The upper figure shows contours on a horizontal cut located 0.9m below the water surface, and the lower figure shows contours of velocity magnitude on a vertical cut through the center of the cage. The incoming velocity was 0.5m s$^{-1}$, and the contours shown are of velocity magnitude divided by 0.5m s$^{-1}$. The mesh used was mesh 2, and the SKE turbulence model was used.

and 0.500m s$^{-1}$. The inlet turbulence quantities were specified as low intensity and low dissipation ($k = 3.75 \cdot 10^{-5}$ and $\epsilon = 2.5 \cdot 10^{-7}$).

Contour plots of velocity magnitude at two cuts through the flow domain of mesh 2 are shown in Figure 4-11. The cuts are horizontal 0.9m below the water surface and vertical at the center of the cage. It can be seen that there is a rather large reduction of the flow both inside and behind the cage. Due to continuity this means that the flow outside this region of reduced flow is faster than the incoming flow. Also it can be noted that the parts of the net that are parallel to the flow create a rather large velocity reduction, which extends quite far behind the end of
Figure 4-12: Contours of pressure distribution at the surface level. The incoming velocity was 0.5 m s$^{-1}$; the mesh used was mesh 2, and the SKE turbulence model was used.

The net. One quite interesting feature to note is the very low mixing of the wake, which seems to continue very far behind the cage without significant spreading.

A contour plot of pressure at the surface is shown in Figure 4-12. In real life this pressure against the ceiling describing the water surface would be represented by a variation in the water level in response to the pressure distribution. The pressure distribution corresponds to the observed velocity distribution, with a deceleration inside and behind the cage due to a positive pressure gradient in the flow direction and an acceleration of the flow in the part of the domain outside the cage and wake due to a negative pressure gradient in the flow direction.

The obtained CFD results for drag force on the cage have been compared in Table 4.4, and the obtained velocities have been compared to each other and the available measured data in Figure 4-13. In Figure 4-13 it can be seen that there is little difference in the results using the different meshes and different thicknesses of porous media. There was a slight effect when including the bottom ring in the simulation. This is best seen in Figure 4-13 (b) and (f), where a drop in velocity is
Figure 4-13: Velocity reduction of the different meshes given in Table 4.4. Figures (a), (b), (c) and (d) are for $u_0 = 0.125 \text{m s}^{-1}$ and figures (e), (f), (g) and (h) are for $u_0 = 0.500 \text{m s}^{-1}$. Figures (a), (b), (c), (e), (f) and (g) are on transects in the tow direction (x-direction). Horizontal and vertical coordinates $(y, z)$ are (a) and (e): $(0, 0.91 \text{m})$, (b) and (f): $(0, 1.37 \text{m})$, (c) and (g): $(1.08 \text{m}, 0.91 \text{m})$. Figures (d) and (h) are on a transect across the tow direction (y-direction) where the tow and vertical $(x, z)$ position is $(4.57 \text{m}, 0.91 \text{m})$. Error bars are based on 1.5% of $u_0$ and 1cm s$^{-1}$ residual current.
Table 4.5: Model settings tested and obtained drag force on the cage. The porous resistance coefficients used are the ones given in Table 4.3 for solidity $S = 0.22$.

<table>
<thead>
<tr>
<th>Model settings</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulence model</td>
<td>SKE</td>
<td>SKE</td>
<td>RKE</td>
<td>SKO</td>
</tr>
<tr>
<td>Discretization order</td>
<td>1</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Drag @ $u_0 = 0.125\text{m s}^{-1}$ (N)</td>
<td>29.33</td>
<td>29.68</td>
<td>27.78</td>
<td>29.63</td>
</tr>
<tr>
<td>Drag @ $u_0 = 0.500\text{m s}^{-1}$ (N)</td>
<td>400.1</td>
<td>406.6</td>
<td>376.2</td>
<td>410.7</td>
</tr>
</tbody>
</table>

seen at x-position 4m to 5m. Otherwise the effect of the bottom ring was negligible. There was some difference in drag force between the different meshes, but there were a lot of uncertainties associated with the drag measurements on the cage, so the drag on the cage was not used to decide on a certain mesh. Mesh 2 (Table 4.4) seemed to produce good data, and the refinement of the region, including the large velocity gradients between the flow in the wake and the surrounding flow, makes this mesh the choice for further investigations.

The effect of using different discretization schemes and turbulence models was also investigated. These simulations were all run using Mesh 2 described in Table 4.4, two velocities ($u_0 = 0.125\text{m s}^{-1}$ and $u_0 = 0.500\text{m s}^{-1}$) and the porous resistance coefficients listed in Table 4.3 for $S = 0.22$. All simulations were run using the inlet turbulence quantities $k = 3.75 \cdot 10^{-5}$ and $\epsilon = 2.5 \cdot 10^{-7}$. The obtained CFD results for drag have been compared in Table 4.5, and the obtained velocities have been compared to each other and the available measured data in Figure 4-14.

The difference between using 1st and 2nd order discretization was small and can only be seen in Figure 4-14 (d) and (h) as a slight increase in the diffusion of the wake. This small effect was probably due to the low diffusion in the hexahedral cells dominating the domain and the diffusive behavior of the 1st order discretization scheme had only a small effect. It can also be seen that the realizable $k - \epsilon$ model (RKE) was much less diffusive than the SKE and that the standard $k - \omega$ model (SKO) seems to be more diffusive than the SKE. The more diffusive models seem to agree best with the data, except for the fact that the velocity reduction behind
Figure 4-14: Velocity reduction of the different model settings given in Table 4.5. Figures (a), (b), (c) and (d) are for $u_0 = 0.125 \text{ms}^{-1}$ and figures (e), (f), (g) and (h) are for $u_0 = 0.500 \text{ms}^{-1}$. Figures (a), (b), (c), (e), (f) and (g) are on transects in the tow direction (x-direction). Horizontal and vertical coordinates (y, z) are (a) and (e): (0, 0.91m), (b) and (f): (0, 1.37m), (c) and (g): (1.08m, 0.91m). Figures (d) and (h) are on a transect across the tow direction (y-direction) where the tow and vertical (x, z) position is (4.57m, 0.91m). Error bars are based on 1.5% of $u_0$ and 1cm $\text{s}^{-1}$ residual current.
the cage was less than what was observed in the data. The drag force was smallest for the RKE and largest for the SKO.

The effect of changing turbulence quantities was also investigated. From the previous tests (Figures 4-13 and 4-14) it seemed that a higher amount of diffusion agrees better with the measurements, and thus a test was made using a higher incoming turbulent kinetic energy, $k$ (justified as residual turbulence in the tank water due to previous runs). The $k$ used for the previous runs was based on a rms value of the turbulent fluctuations $u'_{rms} = 0.005 \text{ms}^{-1}$, and $\epsilon$ was based on a length scale (the size of the energy containing turbulent eddies) $l = 0.15 \text{m}$. This length scale was quite large, which leads to a small dissipation. To test the effect of increasing the incoming turbulence, a new set of $k$ and $\epsilon$ was calculated using a much higher (probably unrealistic) $u'_{rms} = 0.025 \text{ms}^{-1}$ and keeping the same large $l = 0.15 \text{m}$. This combination of a high $k$ and low $\epsilon$ should demonstrate high turbulent diffusion. A test was also performed setting the $k$ and $\epsilon$ values to correspond with what was generated by the twines in the net. Here $k \sim 3.5 \cdot 10^{-4} \text{(m}^2\text{s}^{-2})$ was based on the measurements by Vincent and Marichal (1996) and $\epsilon \sim 1 \cdot 10^{-4} \text{(m}^2\text{s}^{-3})$ was based on a length scale $l = 0.01 \text{m}$ assumed as a approximate length scale for vortices generated by the net twines with a diameter of 2.8mm. These numbers were very approximate, since Vincent and Marichal (1996) only give a dimensionless plot of mean squared velocity fluctuations across the net in a cone, but no other information about the net or velocity. The numbers were thought to be the right order of magnitude, though. The input values for the tests and drag forces obtained are given in Table 4.6, while velocity plots are shown in Figure 4-15.

The results show that increasing the incoming turbulence adds more diffusion to the wake, but the effect was only seen in the outer edge of the wake and in the lowest parts of the cage and wake. The effect seen in Figures 4-15 (b), and (f) were due to the added diffusion of the wake generated by the bottom net. Defining
Figure 4-15: Velocity reduction using different turbulence quantities given in Table 4.6. Figures (a), (b), (c) and (d) are for $u_0 = 0.125\text{m}\text{s}^{-1}$ and figures (e), (f), (g) and (h) are for $u_0 = 0.500\text{m}\text{s}^{-1}$. Figures (a), (b), (c), (e), (f) and (g) are on transects in the tow direction (x-direction). Horizontal and vertical coordinates (y, z) are (a) and (e): (0, 0.91m), (b) and (f): (0, 1.37m), (c) and (g): (1.08m, 0.91m). Figures (d) and (h) are on a transect across the tow direction (y-direction) where the tow and vertical (x, z) position is (4.57m, 0.91m). Error bars are based on 1.5% of $u_0$ and 1cm s$^{-1}$ residual current.
Table 4.6: Turbulence quantities tested, and obtained drag force on the cage. The porous resistance coefficients used are the ones given in Table 4.3 for solidity $S = 0.22$.

<table>
<thead>
<tr>
<th>Turbulence settings</th>
<th>1</th>
<th>2</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k$ at inlet (m$^2$s$^{-2}$)</td>
<td>3.75·10$^{-5}$</td>
<td>9.4·10$^{-4}$</td>
<td>3.75·10$^{-5}$</td>
</tr>
<tr>
<td>$\epsilon$ at inlet (m$^2$s$^{-3}$)</td>
<td>2.5·10$^{-7}$</td>
<td>3.2·10$^{-5}$</td>
<td>2.5·10$^{-7}$</td>
</tr>
<tr>
<td>Constant $k$ inside net (m$^2$s$^{-2}$)</td>
<td>-</td>
<td>-</td>
<td>3·10$^{-4}$</td>
</tr>
<tr>
<td>Constant $\epsilon$ inside net (m$^2$s$^{-3}$)</td>
<td>-</td>
<td>-</td>
<td>1·10$^{-4}$</td>
</tr>
<tr>
<td>Drag @ $u_0 = 0.125$ m s$^{-1}$(N)</td>
<td>29.33</td>
<td>30.34</td>
<td>27.46</td>
</tr>
<tr>
<td>Drag @ $u_0 = 0.500$ m s$^{-1}$(N)</td>
<td>400.1</td>
<td>408.7</td>
<td>374.0</td>
</tr>
</tbody>
</table>

Table 4.7: Porous resistance coefficients tested, and obtained drag force on the cage.

<table>
<thead>
<tr>
<th>Coefficients</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D_n$ (m$^{-2}$)</td>
<td>100100</td>
<td>100100</td>
<td>100100</td>
<td>150150</td>
<td>67067</td>
</tr>
<tr>
<td>$D_t$ (m$^{-2}$)</td>
<td>32360</td>
<td>48540</td>
<td>21681</td>
<td>32360</td>
<td>32360</td>
</tr>
<tr>
<td>$C_n$ (m$^{-1}$)</td>
<td>5.952</td>
<td>5.952</td>
<td>5.952</td>
<td>8.928</td>
<td>3.9878</td>
</tr>
<tr>
<td>$C_t$ (m$^{-1}$)</td>
<td>2.105</td>
<td>3.1575</td>
<td>1.4104</td>
<td>2.105</td>
<td>2.105</td>
</tr>
<tr>
<td>Drag @ $u_0 = 0.125$ m s$^{-1}$(N)</td>
<td>29.33</td>
<td>32.32</td>
<td>26.91</td>
<td>35.43</td>
<td>24.41</td>
</tr>
<tr>
<td>Drag @ $u_0 = 0.500$ m s$^{-1}$(N)</td>
<td>400.1</td>
<td>439.8</td>
<td>367.7</td>
<td>488.5</td>
<td>330.4</td>
</tr>
</tbody>
</table>

The turbulence inside the porous media according to Turb 5 values reduced the turbulent diffusion due to the rather high $\epsilon$ inside the porous media, which boosted turbulent dissipation inside the cage and in the wake.

The effect of varying the porous media resistance coefficients was also investigated. The default values of the coefficients were the ones in Table 4.3, and for the test the resistance coefficients were increased by 50% and decreased by 33%. The coefficients used and the drag forces obtained from the tests are given in Table 4.7, and the velocity plots are shown in Figure 4-16.

The results from varying the porous resistance coefficients show that changing the normal porous resistance coefficients has a much larger effect than changing the tangential resistance coefficients. None of the changes increased the velocity reduction in the wake region without also increasing the velocity reduction inside the cage as well and increasing the drag force to a level much higher than what is seen in Table 4.1. Therefore applying further adjustments to the porous media resistance
Figure 4-16: Velocity reduction using the different porous resistance coefficients given in Table 4.7. Figures (a), (b), (c) and (d) are for $u_0 = 0.125 \text{m s}^{-1}$ and figures (e), (f), (g) and (h) are for $u_0 = 0.500 \text{m s}^{-1}$. Figures (a), (b), (c), (e), (f) and (g) are on transects in the tow direction (x-direction). Horizontal and vertical coordinates (y, z) are (a) and (e): (0, 0.91m), (b) and (f): (0, 1.37m), (c) and (g): (1.08m, 0.91m). Figures (d) and (h) are on a transect across the tow direction (y-direction) where the tow and vertical (x, z) position is (4.57m, 0.91m). Error bars are based on 1.5% of $u_0$ and $1 \text{cm s}^{-1}$ residual current.
coefficients was not expected to improve the results and was found unnecessary.

4.2.3 Discussion

The velocity reduction inside the cage agreed well with data measured inside the cage, but there was a discrepancy between the observations and the simulation results in the wake region. None of the variations of grids, model settings or porous resistance coefficients seemed to make the model results agree better with the data in the wake region.

There can be different explanations for this difference between measurements and model results.

• The turbulence models used applied the isotropic eddy viscosity achieved by the Boussinesq approximation. This approximation might not be accurate enough in this situation, and other turbulence models might provide better solutions. Turbulence models that could be tested in the future include Reynolds stress models and large eddy simulation (LES) models. Because these models require a lot of computational power due to a large 3D domain with small scales that need to be resolved, running these models was regarded as outside the scope of the present work.

• There might be some feature of the flow through the net that was not adequately modeled using the homogeneous porous media model. At the moment there is not enough data available to investigate this further. This would require measuring the small scale flow features around the individual net twines and knots. Another approach to investigate this problem would be to model a small section of the net comparing a model using the real geometry of the net to a model using the porous media model of the net.
• There might be some free surface effects, but the wavelength and waveheight for the surface waves generated is small and probably not noticeable at 0.91m water depth. Specifically, if a surface wave disturbance were to move with the cage at 50cm s\(^{-1}\), the dispersion relation would require a wavelength of 16cm. This "deep water" wave would have negligible effect on velocity measurements at a depth of 91cm. Therefore, it is difficult to believe that the free surface effects account for more than a small portion of the error.

• There is, of course, also a slight possibility that there is something wrong with the measurements. This can only be tested by comparing to similar experiments.

The above mentioned possible explanations for the difference between measured and modeled data should all be explored in further work in this field of research.

4.3 USNA Cage at "Full Scale"

The cage studied in Sections 4.1 and 4.2 was a lot smaller than cages used in the commercial aquaculture of salmonids which was the primary focus of this study. A computer modeling study was performed in which the small USNA cage was scaled up to a size that is common in this industry. Cage diameter was increased from 3.124m to 30m, and the domain was scaled with the same scale factor. CFD simulations of the flow through and around the cage were performed and compared to the simulations in Section 4.2.

4.3.1 Thickness Test

When modeling the full size cage, the thickness of the porous media representing the net played a crucial role in the number of grid cells that were needed for the
Figure 4-17: Comparing nets of different thickness parallel to the flow. The solid line is for porous media thickness 5cm; dashed line, 10cm; and dash-dot line, 30cm. y-coordinate (horizontal axis of plot) is across the flow and the net is placed along the x-coordinate stretching from -15m to 15m.

mesh. The smallest cells in the domain were those inside the porous media, and they were limited in size to a maximum cell size similar to the thickness of the porous media or a little bit larger. For this reason, the effect of increasing the thickness of the porous media in a 30m long, 2D net panel oriented parallel to the flow was investigated. This was the only orientation of the net panel used in the test, since at other orientations, the across flow dimension of the net panel would be much larger than the thickness of the porous media, and the effect of the various thicknesses would be negligible. Three different thicknesses were investigated - 5, 10 and 30cm, using the porous resistance coefficients given in Table 4.8. The resistance coefficients for the larger thicknesses were found by multiplying the coefficients for the 5cm thick porous media with the ratio between the thicknesses. The drag forces on the net panels are also given in Table 4.8. Velocity profiles across the flow from the center of the net panel to 3m out are shown in Figure 4-17. Four profiles are shown for each of the porous media thicknesses, the first at the start of the porous media ($x = -15m$) and the last, 5m behind the porous media ($x = 20m$).
Table 4.8: Porous resistance coefficients used for the thickness test, and obtained drag force on the porous media.

<table>
<thead>
<tr>
<th>Thickness (cm)</th>
<th>5</th>
<th>10</th>
<th>30</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D_n$ (m$^{-2}$)</td>
<td>100100</td>
<td>50050</td>
<td>16683</td>
</tr>
<tr>
<td>$D_t$ (m$^{-2}$)</td>
<td>32360</td>
<td>16180</td>
<td>5393</td>
</tr>
<tr>
<td>$C_n$ (m$^{-1}$)</td>
<td>5.952</td>
<td>2.976</td>
<td>0.9920</td>
</tr>
<tr>
<td>$C_t$ (m$^{-1}$)</td>
<td>2.105</td>
<td>1.0525</td>
<td>0.3808</td>
</tr>
<tr>
<td>Drag @ $u_0 = 0.500$ m s$^{-1}$ (N)</td>
<td>141.1</td>
<td>171.1</td>
<td>152.5</td>
</tr>
</tbody>
</table>

The results indicated that there was not a large difference between the three tested thicknesses. There was some difference in drag force and a very slight difference in velocity reduction. The drag force on the parts of the net in a fish cage that are oriented parallel to the flow is only a small portion of the total drag force. Therefore the differences in drag force observed in Table 4.8 were not considered a problem, and all of the tested thicknesses should work well for a full size cage that generally is built using diameters of at least 30m.

4.3.2 Scaled-up USNA Cage Model

At the start of the full scale modeling, a model was built using dimensions representing a similar setup to the one for the USNA cage (Section 4.1) but scaled up such that the cage had a diameter of 30m. The scale factor was 9.603. The scaled up dimensions of the domain were: length = 480m, width = 76m and depth = 47m. The cage had a depth of 16.2m and was positioned 96m from the inlet end of the domain. To keep the number of grid cells reasonably low, a porous media thickness of 30cm was used. The boundaries were specified the same way as for the models in Section 4.2. The grid was created using the TGrid meshing scheme inside the porous media and the Hex Core meshing scheme for the fluid inside and outside the cage. The cell size inside the porous media was 40cm. This was used as the minimum cell size, and the cell size in the rest of the domain increased away from the porous media with a growth rate of 1.2 up to a maximum size of 5m. Using
Figure 4-18: Velocity in scaled up cage using three sets of inlet turbulence parameters. The inlet velocity is $0.500\text{ms}^{-1}$. Figures (a), (b) and (c) are on transects in the tow direction (x-direction). Horizontal and vertical coordinates $(y, z)$ are (a): $(0, 8.7\text{m})$, (b): $(0, 13.2\text{m})$ and (c): $(10.4\text{m}, 8.7\text{m})$. Figure (d) is on a transect across the tow direction (y-direction) where the in-tow and vertical $(x, z)$ position is $(43.9\text{m}, 8.7\text{m})$.

This cell size specification and the Hex Core meshing scheme proved successful, and the number of grid cells was as low as 657000.

The simulations were performed using the $1^{st}$-order discretization scheme and the SKE turbulence scheme. The porous resistance coefficients were the ones given in Table 4.8 for a 30cm thick porous media. A rather wide range of inlet turbulence parameters was chosen. The same $k$ and $\epsilon$ as used for the tank test validation was the low end of the parameters. This was unrealistically low for tidal currents, and two more cases were tested using 2% and 10% turbulence intensity with length scales of 0.15m and 5m respectively. The values for $k$ and $\epsilon$ for the turbulence cases are found in Table 4.9.

Figure 4-18 contains plots of velocity magnitude along similar transects as used
Table 4.9: Turbulence quantities tested, and obtained drag force

<table>
<thead>
<tr>
<th>Turbulence</th>
<th>low</th>
<th>med</th>
<th>high</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k$ at inlet (m$^2$s$^{-2}$)</td>
<td>$3.75 \cdot 10^{-5}$</td>
<td>$1.5 \cdot 10^{-4}$</td>
<td>$3.75 \cdot 10^{-3}$</td>
</tr>
<tr>
<td>$\varepsilon$ at inlet (m$^2$s$^{-3}$)</td>
<td>$2.5 \cdot 10^{-7}$</td>
<td>$2.01 \cdot 10^{-6}$</td>
<td>$7.55 \cdot 10^{-6}$</td>
</tr>
<tr>
<td>Drag @ $u_0 = 0.500$ m s$^{-1}$ (kN)</td>
<td>35.55</td>
<td>35.81</td>
<td>40.90</td>
</tr>
</tbody>
</table>

in Figures 4-13, 4-14, 4-15 and 4-16, but all dimensions are scaled up using the scale factor for the cage (9.603). The model was only run for $u_0 = 0.50$ m s$^{-1}$. The drag force on the cage is provided in Table 4.9.

The results (Figure 4-18) show that the velocity reduction along the transects is quite similar to the reduction observed in Section 4.2, except that a higher diffusion of the wake is observed at the higher turbulence levels. It is interesting to see that the two lower turbulence levels produced almost the same result. When comparing these results to the USNA cage results regarding the effect of scaling from the small cage to the full size cage, the results for Mesh 1 in Figure 4-13 (e) to (h) should be used. It can be seen that there was higher diffusion of the wake in the full scale model. This can be explained by the higher turbulence levels generated by the full scale cage and that Mesh 1 in Figure 4-13 was modeled using 2$^{nd}$ order discretization while the full scale cage was modeled using 1$^{st}$ order discretization. Calculating the drag coefficients for the low turbulence case in Table 4.9 and Mesh 1 in Table 4.4, using the projected outline area of the cage, gave $C_d = 0.577$ for the cage in the tank test and $C_d = 0.585$ for the full scale cage. The difference was less than 2%, so the effect of scaling on the drag coefficient was small. The presented results (e.g. Table 4.6 and 4.9) indicate that increasing the turbulence levels tended to increase drag. The full scale cage produced higher turbulence levels than the small cage, which might explain the slight increase in drag coefficient.
4.4 Study of a Commercial Cage

A full size cage common in the salmon farming industry of the Faroe Islands was modeled. The cage was circular and flat bottomed with a diameter of 30.6m and a depth of 11m. Measurements of current reduction inside and behind such a cage moored at a high current aquaculture site at Gulin, Faroe Islands (Patursson and Simonsen, 2008) were compared to the model results.

4.4.1 Field Measurements

The cage was moored at a site experiencing relatively strong tidal currents in 30m of water. No other cages were on the site. Measurements of current reduction were performed when the incident current speed was around 50cm s\(^{-1}\). The current measurements were performed using a single point electro-magnetic current meter (AEM-HR, from Alec Electronics Co., Ltd) inside the cage and a down-looking, boat-mounted ADCP (300kHz Workhorse Sentinel from Teledyne RD Instruments) with bottom tracking software to measure the current in front and in the wake of the cage. The ADCP available for the measurements was a low frequency (300kHz) ADCP, and with the chosen bin size (1m) the vendor supplied standard deviation of the current measurements for the ADCP was 12.6cm s\(^{-1}\). Due to this large standard deviation, measurements at slower speeds were not used. The ADCP measurements consisted of vertical profiles through the water column, from 3m below the surface down to a couple of meters above the bottom. Profiles were taken across the flow in front of the cage to measure incoming flow and behind the cage to measure reduced current. The raw datapoints were processed to obtain the average of a representative piece of the profile down to 10m below the surface. The measurement in front of the cage was approximately one cage diameter in front of the cage and was used as the undisturbed velocity. The measurement inside the
cage was centered in the cage, and the measurement in the wake was approximately one cage diameter behind the cage. The measurements inside the cage and behind the cage were compared to the measurement in front of the cage to obtain velocity reduction for these positions. The velocity reduction results were 26% inside the cage and 42% behind the cage.

The net material was nylon; mesh size was 25mm (barlength), and twine thickness was 2mm. The net had only been in the water for a few weeks, so regular biofouling was very limited, but the net was fouled by a large number of jellyfish during the time of the measurements. The jellyfish were carried by the current and were mostly attached to the upcurrent net, so it is fair to assume that the drag coefficient of the upcurrent net was larger than the drag coefficient of the downcurrent net.

4.4.2 CFD Model

The cage was modeled using the actual dimensions of the cage with a 30cm thick porous media to describe the net. The domain was 500m long, 200m wide and 35m deep. The flow followed the long dimension of the domain, and the cage was positioned 100m away from the inlet. An overview of the cage and domain is shown in Figure 4-19. The bottom of the domain was modeled as stationary wall with a no-slip boundary condition, while the surface and sides were modeled as stationary walls with no shear stress. The inlet face was a velocity inlet with a constant velocity, and the outlet was a pressure outlet.

The cases tested were a clean net and a biofouled net. The porous media resistance coefficients used for the clean net were the ones defined in Table 4.8. To find the porous resistance coefficients for the biofouled net, some choices had to be made. Swift et al. (2006) measured drag coefficients of biofouled net panels at normal incidence. It was chosen to base the coefficients on the highest measured
drag coefficient from Swift et al. (2006), which was 0.599 for a net panel with a blockage of 0.566. This particular net panel was fouled by a distribution of hydroids that was thought to have the same flow resistance as the jellyfish on the Faroe Islands net. Three different ways of assessing the porous resistance coefficients from this drag coefficient were considered:

- The porous resistance coefficients for the clean net panel could have been increased through an iterative process until the drag coefficient at normal angle of attack reached the desired value ($C_d = 0.599$).

- The blockage of the biofouled net could have been used together with Figure 4-8 to find the coefficients. The blockage measured by Swift et al. (2006) was much higher than any of the nets in Figure 4-8 which would have given unrealistically high coefficients.

- The porous resistance coefficients from the net with the drag coefficient closest to 0.6 in Figure 3-16 could have been used.

Since both the maximum drag coefficient measured by Swift et al. (2006) and the drag coefficient at normal angle of attack at the highest measured Re for a net with solidity 0.317 from Rudi et al. (1988) (Figure 3-16) were around 0.6, using the porous resistance coefficients for this net seemed like a good choice. The only difference was that Swift et al. (2006) assumed a constant drag coefficient with
Table 4.10: Porous coefficients for the biofouled net. The coefficients were from the fit to data from Rudi et al. (1988) in Table 3.12.

<table>
<thead>
<tr>
<th>t (cm)</th>
<th>$D_n$ (m$^{-2}$)</th>
<th>$D_t$ (m$^{-2}$)</th>
<th>$C_n$ (m$^{-1}$)</th>
<th>$C_t$ (m$^{-1}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>845370</td>
<td>105900</td>
<td>12.401</td>
<td>6.5737</td>
</tr>
<tr>
<td>30</td>
<td>140895</td>
<td>17650</td>
<td>2.0668</td>
<td>1.0956</td>
</tr>
</tbody>
</table>

Re while the drag coefficient measured by Rudi et al. (1988) was larger than 0.6 at small Re and decreased with increasing Re down to about 0.6. This meant that using this set of porous coefficients might overpredict the drag and hence velocity reduction inside and behind the net at small velocities. On the other hand, this set of coefficients was based on measurements and found by fitting the function explained in Section 3.3.2 to drag and lift force measurements, which was an advantage over the other two approaches mentioned. The porous resistance coefficients for the biofouled net modeled using a 30cm thick porous media were therefore found by multiplying the coefficients for the net with solidity 0.317 in Table 3.12 by the ratio $\frac{5}{30}$ since the coefficients in Table 3.12 were found for a 5cm thick porous media. The coefficients for the biofouled net are provided in Table 4.10.

The porous media was meshed using the TGrid meshing scheme with a cell size of 0.4m. The volume inside the cage was meshed using the Hex Core meshing scheme, and the part of the domain outside the cage was meshed using the TGrid scheme. The cell size increased by a factor of 1.2 away from the net both inside and outside the cage to a maximum size of 5m. The total number of grid cells was 718000. The mesh is shown in Figure 4-20. The reason for using the TGrid scheme for the large part of the domain outside the cage was due to the fact that the cell size was large compared to the depth of the domain. Thus the number of hexahedral cells created by the Hex Core scheme would have been very limited or none at all. The simulations were run using a 10% turbulence intensity with a length scale of 5m and a 2% turbulence intensity with a length scale of 0.15m. This
Figure 4-20: The mesh for the full size cage and domain. The uppermost figure is looking down on a horizontal cut through the domain at the middle depth of the cage. The lowest figure is a side view of a vertical cut through the center of the domain. The two middle figures are closeups of of chosen sections of the domain, where the porous media is given a gray color.

Table 4.11: Turbulence in full scale.

<table>
<thead>
<tr>
<th>Speed (cm s(^{-1}))</th>
<th>Low turbulence</th>
<th>High turbulence</th>
</tr>
</thead>
<tbody>
<tr>
<td>12.5</td>
<td>50</td>
<td>12.5</td>
</tr>
<tr>
<td>(k) at inlet (m(^2)s(^{-2}))</td>
<td>9.38(\times)10(^{-6})</td>
<td>1.50(\times)10(^{-4})</td>
</tr>
<tr>
<td>(\epsilon) at inlet (m(^2)s(^{-3}))</td>
<td>3.14(\times)10(^{-8})</td>
<td>2.01(\times)10(^{-6})</td>
</tr>
</tbody>
</table>

gave the inlet turbulence quantities in Table 4.11.

4.4.3 Flushing Time

The flushing time was found by calculating the incoming flow through the cage surface. This was, of course, also equal to the exit flow under steady state conditions. The flushing time was then determined by
Flushing time = \frac{\text{Volume of cage}}{\text{Rate of flow of water into the cage}} = \frac{V}{Q} \quad (4.2)

The volume of the cage was calculated using the center of the porous media as the outer surface. This meant that the volume of the cage was calculated as a cylindrical volume with diameter \( d = 30.6 \text{m} \), and axial length \( l = 11 \text{m} \). This gave volume \( V = 8090 \text{m}^3 \). The rate of flow of water into the cage was calculated as the flow entering (or exiting) through the cage wall integrated over the surface where the flow entered (or exited). Thus rate of flow was computed as

\[ Q = \int \vec{u} \cdot \vec{A} dA \quad (4.3) \]

where \( \vec{u} \) was the flow through the porous media, and \( \vec{A} \) was the area vector that was always perpendicular to the surface of integration. The discrete form was

\[ Q = \sum \vec{u}_i \cdot \vec{A}_i \quad (4.4) \]

where \( \vec{u}_i \) was \( \vec{u} \) through face \( i \), and \( \vec{A}_i \) was the area vector perpendicular to face \( i \) with the length defined as the area of face \( i \).

A simplified, approximate calculation of \( Q \) was also done assuming that the flow inside the cage was in the same direction as \( u_0 \) and was spatially uniform. Then rate of flow was calculated as the average speed inside the cage times the projected area of the cage, so that

\[ Q \approx (u_{\text{inside}}) A_p \quad (4.5) \]
Figure 4-21: Velocity in the full size cage using the two sets of inlet turbulence parameters explained in Table 4.11 and the two sets of porous resistance coefficients in Table 4.10. The two sets of porous resistance coefficients used can be distinguished by the color of the lines, and the different sets of inlet turbulence parameters are distinguished by line type. Low turbulence was plotted using a solid line, and high turbulence was plotted using a dashed line. Figures (a) and (b) are for $u_0 = 0.125 \text{m s}^{-1}$ and figures (c) and (d) are for $u_0 = 0.500 \text{m s}^{-1}$. Figures (a) and (c) are on a transect in the fluid flow direction (x-direction) through the center of the cage at 5m water depth. Figures (b) and (d) are on a transect across the tow direction (y-direction) 45m behind the center of the cage. The plot starts at the centerline of the wake.

4.4.4 Results

The velocity distributions obtained by the simulations are shown in Figure 4-21. The plots shown are velocity magnitude along the transects seen in Figure 4-19. One was in the flow direction through the center of the cage at a depth of 5m, and the other was across the flow 45m behind the cage center. The data from the field measurements is also shown in the transect along the centerline of the cage at $u_0 = 0.5 \text{m s}^{-1}$.
The results show that the velocity reduction for the clean net was approximately the same as for the simulations of the USNA cage (Section 4.2). The velocity reduction for the biofouled net was, however, much larger than for the clean net and was also much larger at the slow speed than at the high speed. This was due to the fact that the drag data used to find the porous resistance coefficients for the biofouled net was for a net with a drag coefficient that decreased with increasing speed. It can also be seen that the higher inlet turbulence led to more diffusion of the wake, and using the higher inlet turbulence, the velocity started to increase 30m behind the center of the cage. The data from the field measurements was very close to model results for the biofouled net. This makes sense since the net in the field tests was fouled by a very large number of jellyfish. It can also be seen that the measurement inside the cage was very close to the model result, but in the wake region the model predicted a larger reduction than the measurements. This makes sense since most of the jellyfish were on the upcurrent part of the cage.

Next the flushing time of the cage was calculated using different approaches for the clean and biofouled nets and the different turbulence quantities where applicable. The flushing time was calculated using five different approaches:

1. The predicted, spatially varying velocity through the net was used and integrated over the part of the cage surface that has flow into the cage using Equation 4.4.

2. The average velocity inside the cage from the CFD results was used with Equation 4.5.

3. The velocity inside the cage was assumed reduced by the effect of one layer of netting according to Equation 3.26 with \( r = 0.412 \) and \( C_d \) for the clean net as measured in Section 3.1 and for the biofouled net as measured by Rudi et al. (1988). This average velocity was then used in Equation 4.5. For the clean net
Table 4.12: Numerical results from simulations

<table>
<thead>
<tr>
<th></th>
<th>Low turbulence</th>
<th>High turbulence</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_0$ (cm s$^{-1}$)</td>
<td>12.5</td>
<td>12.5</td>
</tr>
<tr>
<td></td>
<td>50</td>
<td>50</td>
</tr>
<tr>
<td>Drag force (kN)</td>
<td>1.790 25.05</td>
<td>2.029 28.41</td>
</tr>
<tr>
<td>$\langle u_{cage} \rangle$ (cm s$^{-1}$)</td>
<td>10.49 43.19</td>
<td>10.50 43.20</td>
</tr>
<tr>
<td>Flushing time$^1$ (s)</td>
<td>231.2 56.31</td>
<td>229.9 56.08</td>
</tr>
<tr>
<td>Flushing time$^2$ (s)</td>
<td>229.1 55.65</td>
<td>228.9 55.63</td>
</tr>
<tr>
<td>Flushing time$^3$ (s)</td>
<td>216.0 53.76</td>
<td>-</td>
</tr>
<tr>
<td>Flushing time$^4$ (s)</td>
<td>227.6 56.89</td>
<td>-</td>
</tr>
<tr>
<td>Flushing time$^5$ (s)</td>
<td>192.3 48.07</td>
<td>-</td>
</tr>
</tbody>
</table>

Clean net

<table>
<thead>
<tr>
<th></th>
<th>Low turbulence</th>
<th>High turbulence</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_0$ (cm s$^{-1}$)</td>
<td>12.5</td>
<td>12.5</td>
</tr>
<tr>
<td></td>
<td>50</td>
<td>50</td>
</tr>
<tr>
<td>Drag force (kN)</td>
<td>3.153 42.53</td>
<td>3.660 50.09</td>
</tr>
<tr>
<td>$\langle u_{cage} \rangle$ (cm s$^{-1}$)</td>
<td>6.86 35.20</td>
<td>6.96 35.21</td>
</tr>
<tr>
<td>Flushing time$^1$ (s)</td>
<td>332.7 67.62</td>
<td>325.3 67.03</td>
</tr>
<tr>
<td>Flushing time$^2$ (s)</td>
<td>350.3 68.28</td>
<td>345.3 68.26</td>
</tr>
<tr>
<td>Flushing time$^3$ (s)</td>
<td>309.2 64.92</td>
<td>-</td>
</tr>
<tr>
<td>Flushing time$^4$ (s)</td>
<td>266.6 66.66</td>
<td>-</td>
</tr>
<tr>
<td>Flushing time$^5$ (s)</td>
<td>192.3 48.07</td>
<td>-</td>
</tr>
</tbody>
</table>

Biofouled net

The measured drag coefficients were 0.267 and 0.257 for 12.5 and 50 cm s$^{-1}$, respectively, and for the biofouled net $C_d$ was 0.918 which was measured at $u_0 = 15.9$ cm s$^{-1}$ and 0.63 which was found from a manually made curve fit to the data.

4. The velocity inside the cage was assumed reduced by the effect of one layer of netting according to the Equations 1.4 and 1.6 by Loland (1991) using $S = 0.22$ for the clean net and $S = 0.317$ for the biofouled net. This average velocity was then used in Equation 4.5.

5. The velocity was assumed undisturbed by the net and was used in Equation 4.5.

The results are shown in Table 4.12.

In Table 4.12 it can be seen that the cases with higher inlet turbulence produce a larger drag force and a shorter flushing time. This makes sense due to the larger mixing of the flow with the higher turbulence levels. There is a difference between
calculating the flushing time with method 1 and method 2 for the biofouled net. This makes sense since the alignment of the flow to the incident velocity was less with larger resistance in the porous media.

When calculating the flushing time with the analytical methods, they produced various results. Method 3, which was the most comprehensive including a measured drag coefficient, produced a lower flushing time than method 1, but the error was consistent. Method 4 produced quite accurate results except for the slow biofouled run, which was due to the fact that $C_d$ calculated by Equation 1.4 was constant with $u$. Method 5, of course, overpredicted flushing, since no reduction in velocity was assumed. The $r = 0.46$ used by Løland (1991) was partly based on tow tests with small square cages, and might be more appropriate than $r = 0.412$ based on the single net panel measurements and simulations in Sections 3.1 to 3.3.
Computational fluid dynamic (CFD) modeling was used to model the flow through and around net panels and cages using a porous media model to describe the net. The model was calibrated using measurements from tow tank experiments. A net panel was towed at different velocities and angles of attack, and drag force and lift force on the net panel and velocity reduction behind the net panel were recorded. The modeling approach was validated using tow tank measurements of drag force on a small gravity cage and velocity reduction inside the cage and in the wake region.

During calibration, the best porous resistance coefficients were obtained by minimizing the difference between CFD predictions and measurements. A data fitting procedure was used where simplified forms of the model equations were used to predict the measured data, and the porous resistance coefficients that gave closest match to the data were found. The four porous resistance coefficients were obtained through the use of the data fitting procedure by iteratively minimizing the error between calculations and data. The experiments used for the calibration made use of a knotless nylon net commonly used in fish farming. The net had a solidity of 0.20 and was tested in a square mesh orientation. The measurements were performed as a part of this study and the measurement setup was designed for the purpose. The method for obtaining the porous media resistance coefficients also made use of test data from Rudi et al. (1988). The CFD method was able to reproduce the drag coefficient and lift coefficient of the net panel and the velocity reduction behind
the net panel with satisfactory accuracy. The largest error was associated with the
velocity reduction at small angle of attack, but this was mostly due to insufficient
modeling of the wake from the net panel pipe frame.

The validation of the CFD method used tow tank measurements involving a
small size gravity cage (3.124m in diameter) performed as a part of this study.
The net used for the validation study was the same net as used for the net panel
measurements, but the solidity was higher (0.22) due to less stretching of the net.
The porous resistance coefficients were based on the coefficients found from the net
panel tests, but increased according to the increased solidity and the data by Rudi
et al. (1988). The modeled drag force was higher than the measured drag force, but
there was an error associated with the measured drag force, so the validation was
based more on comparing current velocity. The modeled current compared well to
the measured current inside the cage, but the reduction was underpredicted in the
wake of the cage.

The turbulence models used were based on an isotropic eddy viscosity. These
models are fairly simple and computationally inexpensive, but the assumption of
using the isotropic eddy viscosity might not be accurate. The model of the net
using the homogeneous porous media was a simplified model where some of the
effects from the net might disappear. One example of a flow feature that was not
included in the model was the effect of the net strands on generating turbulence.
Also the assumption of using the rigid water surface was a limitation that might
affect the results. Finally, it is also important to note that only one measurement
series was used for the validation and there is always a slight chance that some error
was associated with the measurement. Topics for further work are summarized in
Section 5.2.

Full scale simulations were performed for a cage with the clean net used for
the validation study and a biofouled net. The predictions were compared with
field measurements of velocity near and within a cage fouled with jellyfish. The measured data compared well with the modeled data for the biofouled net. Flushing rates were calculated for the clean and the biofouled net from the simulation results and using some analytical approaches. The analytical approaches seemed capable of giving reasonably accurate results. When the net was changed from clean to biofouled, flushing time increased by 41 - 44% for $u_0 = 0.125 \text{m s}^{-1}$ and by 20% for $u_0 = 0.500 \text{m s}^{-1}$. Drag force increased by 70 - 80% for both speeds.

5.1 Overall Conclusions

- The well known equations for modeling flow through porous media have been used together with the Reynolds averaged Navier-Stokes equations to model the flow through and around net panels and fish farming cages. The net was modeled as a thin volume of porous media.

- A method for finding the porous resistance coefficients was developed using simplified versions of the equations solved in the model. The method was successful in finding appropriate coefficients. For the used net, the coefficients $C_n$ and $C_t$ had much larger influence on the error between predicted and measured data than $D_n$ and $D_t$.

- The relation between solidity and porous resistance coefficients was investigated in Section 4.2.1. For the data available there was no clear relationship between solidity and porous resistance coefficients, and more work is needed to find a relation between net geometry and porous resistance coefficients.

- The modeling approach was validated using tow tank measurements of drag force for a small gravity cage and current speed inside and behind the cage. Inside the cage the agreement between predictions and measurements of cur-
rent speed was very good, while the speed in the wake region and the drag force were slightly overpredicted.

- Using different turbulence models had an effect on the predictions. During the validation work, it was not possible to decide whether one of the turbulence models was superior to the other models. Additional data on the mixing of the far wake of fish farming cages would give a better base for deciding between turbulence models.

- Using different inlet turbulence parameters had an effect on the results. Higher turbulent energy and lower dissipation at the inlet induced increased mixing of the wake behind the cage.

- Comparing predicted drag force and flushing time for a full size gravity cage using a clean and a biofouled net showed that even if the drag force increased by up to 80%, the flushing time only increased by up to 40%. The porous resistance coefficients used for the biofouled net were an estimate, and a better knowledge about forces acting on biofouled nets is needed.

### 5.2 Future Work

There are some further investigations that could contribute towards additional validation of the modeling approach. The difference between the model predictions and the measured velocity reduction in the wake of the small gravity cage used for the validation is of special concern. Effects that could clarify this issue, but were outside the scope of the present work, are summarized below:

- To exclude that there was something wrong with the measured data, the model should be validated using some other measurements on a similar structure.
• Higher order turbulence models should be applied, such as Reynolds stress models or large eddy simulation models.

• Further measurements are needed in the far wake of a cage to validate the rate of diffusion of the wake. This is of high importance when several cages in a group are modeled and when cages are positioned in the wake of other cages.

• A free surface model should be tested to investigate the effect of the rigid surface used in the present work.

• A detailed study of the effect of modeling a net as a homogeneous porous media should be performed. This could be done by comparing CFD model results to very detailed measurements of the flow through a net panel (e.g. PIV measurements).

When using the modeling approach for practical applications, a methodology is needed to specify the porous resistance coefficients without performing an extensive measurement program on the net used. The relationship between the porous resistance coefficients and solidity of the net panels tested in Section 3.3.2 is shown in Figure 4-8, but there does not seem to be a simple relationship. Further investigations regarding a method for obtaining porous resistance coefficients would contribute to the usability of the modeling approach.

The effect of biofouling on the nets of fish cages is not well known, and studies are needed to expand on the work by Swift et al. (2006). A special concern is that biofouling comes in many different sizes and shapes, that interact differently with the flow, and detailed studies are needed - e.g. tow tank studies including artificial biofouling.
5.3 Outlook

The objective with the present work was to gain more knowledge about the flow through fish farming cages and to develop a methodology that can be used for other structures of net. The method seems to give reasonably accurate results, at least inside the cage, so the primary objective has been achieved, but as stated above there is still more work to be done.

There are many areas in which the present approach can be helpful. Using this method, a better input of current velocity can be applied in calculations for design of aquaculture cages and moorings. The method can also be applied to model the oxygen distribution and distribution of effluents within a group of cages. In the fisheries industry, knowing the flow distribution inside trawls is of high importance. Testing the present approach on such structures would be an interesting extension of the present work.


