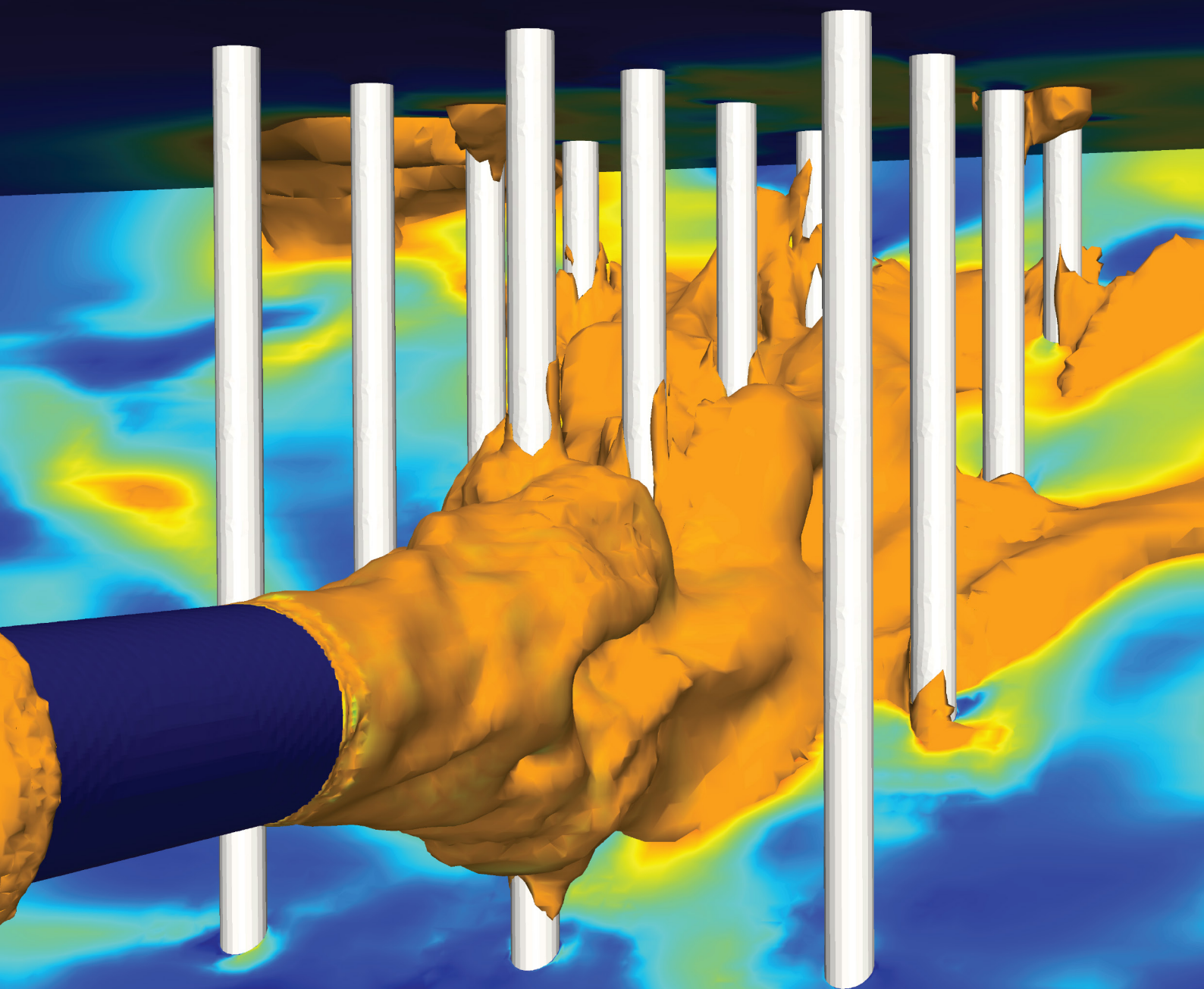


MSc Thesis

NUMERICAL MODELLING OF BOW THRUSTERS AT OPEN QUAY STRUCTURES

Jurjen de Jong

January 2014



MSc Thesis

NUMERICAL MODELLING OF BOW THRUSTERS AT OPEN QUAY STRUCTURES

Jurjen de Jong

January 2014

GRADUATION COMMITTEE

Graduation Professor

Prof. ir. T. Vellinga (Ports and Waterways, TU Delft)

Daily Supervisor

Ir. H.J. Verheij (Ports and Waterways, TU Delft)

Ir. T. Blokland (Ingenieursbureau van de Gemeente Rotterdam)

Dr. ir. H.J. de Koning Gans (Ship Hydromechanics & Structures, TU Delft)

Dr. ir. R.J. Labeur (Environmental Fluid Mechanics, TU Delft)

ABSTRACT

Bow thrusters are of great help for the navigation at quay walls, but the high and turbulent velocities can result in a bed load exceeding the strength of the bed or bed protection. To be able to design a stable bed the velocities at the bed need to be accurately determined. In design practise the velocities generated by a propeller are determined with formulae based on a mix of the momentum theory and measurements. The application of the formulae is often limited to cases for which measurements have been carried out and do not allow a secure design for more complicated structures and the different velocity field of a bow thruster.

To improve the calculation of velocities on a slope, a large number of measurements were done by Van Doorn [TU Delft, 2012] for several scenarios with and without piles and resulted in an amplification of the design formula for some of his scenarios. To also predict the velocities for other scenarios these measurements are used to build and calibrate a numerical model.

The open source CFD package OPENFOAM is used for the construction of this numerical model. As the implementation of a rotating propeller in the mesh will result in high computational costs and to allow a fine calibration of the propeller efflux, the propeller is simplified to an actuator disc. At the actuator disc an axial and tangential body force, varying over the radius, are added to the momentum equations in the OPENFOAM solver. Functions for both a ducted and a free propeller are simulated and show comparable results, the free *Goldstein* propeller functions are further applied. The coefficients are estimated based on the measured thrust and torque and calibrated to achieve a good fit to the measured efflux. A local increase of the turbulence at the hub and the propeller tip is not implemented in the numerical computations.

In the sensitivity analysis it is shown that the efflux of the actuator disc is stable for changes to the disc thickness, disc location or local mesh. It is, however, influenced by the chosen turbulence model and it is shown that a Large Eddy Simulation (LES) results in more accurate velocities estimations than the earlier applied RANS model. Due to the long simulations runs of LES simulations, this can only be applied to a limited number of simulations. Changing the mesh at the slope or changing the (rough) wall function of the slope also influences the velocities on the slope to a great extend. Further research is necessary to substantiate a choice for any of these wall approximations.

Comparing the calibrated model to the measured diffusion in axial direction, shows a very good agreement and the numerical model nearly exactly computes the distribution as derived

by Blaauw and van der Kaa. When comparing the velocities at the slope to the derived velocities by Blokland, the numerical model performs well for gentle slopes (1:2.5), but underestimates the velocities for steeper slopes (1:1.5). Also in the physical scale model, higher velocities are measured for the steeper slopes at a slightly lower location and at the toe of the slope.

The addition of piles at the slope results in a small increase of the velocities (5%) for locations in between the piles. At locations closer to the piles a higher increase is visible (35%). These maximum velocities at the piles agree very well with the scale model measurements.

At a vertical quay wall the bottom velocities are underestimated in the numerical model but show the expected reduction to the velocities for increasing distances to the quay wall. For oblique walls the downward water jet, as described by the equation of Römisch, is compared to the numerical simulations. It shows a good comparison for angles up to 30 degree, but for higher angles the downward velocities reduce and the quay will show more comparable behaviour to a slope with only upward velocities.

Up scaling the model to a full-size simulation shows that the dimensionless velocities do not change with a changing scale. This confirms that the model can also be applied to geometries in reality. It is concluded that for a model that is calibrated to a correct (measured) efflux, the flow velocities at structures on (gentle) slopes can be accurately simulated.

ACKNOWLEDGMENTS

This report is the final report of a MSc thesis project to the flow velocities induced by bow thrusters. It involves a research in which scale model measurements, that were carried out by another student at the TU Delft, are used for the creation of a numerical model to better predict the flow velocities near open quay structures.

The research was carried out at Deltares, where resources were available for the running of the simulations. At Deltares I experienced a healthy working environment, helping to generate the motivation when results were strange or disappointing. I would like to express my gratitude to all fellow students at the RIV-department that helped making it a pleasant time with many insightful exchanges of thoughts.

Also words of thanks to my graduation committee consisting of prof. ir. T. Vellinga, ir. H.J. Verheij, ir. T. Blokland, dr. ir. H.J. de Koning Gans and dr. ir. R.J. Labeur. Each member had his share in the committee from a different background. This resulted in many opinions and useful feedback on both the context, as well as the on the structure of this report.

At last I would like to show my gratitude for all friends and family, who have supported me during my study.

Delft, January 2014

Jurjen de Jong

TABLE OF CONTENTS

1	INTRODUCTION	1
1.1	Erosion induced by bow thrusters	1
1.2	Objective and research questions.....	3
1.3	Research methodology	3
2	THEORETICAL BACKGROUND	5
2.1	Physics of a bow thruster.....	5
2.2	Approximating the propeller with an actuator disc	8
2.2.1	Free propellers with the Goldstein optimum	8
2.2.2	Ducted propellers.....	11
2.3	Flow velocities in design approaches.....	14
2.3.1	Efflux velocity	14
2.3.2	Flow field of an unconfined jet.....	15
2.3.3	Velocities at a slope	16
2.3.4	Velocities at a vertical wall	18
2.3.5	Velocities at an oblique wall.....	18
2.3.6	Velocities at piles	19
2.4	Erosion	20
2.5	Conclusion - Increase in accuracy necessary	20
3	MEASUREMENTS AND EARLIER NUMERICAL MODELS	21
3.1	Measurements at open quay structures	21
3.1.1	Efflux velocities and turbulence.....	23
3.1.2	Calculation of the thrust and torque.....	24
3.1.3	Change in time	26
3.2	Measurements at quay walls.....	27
3.3	Evaluation of former numerical models	28
3.4	Conclusion - Necessities numerical model	30

4	APPLICATION OF A NUMERICAL MODEL	31
4.1	OpenFOAM explained	31
4.2	Implementation of the bow thruster	32
4.2.1	Modelling of the propeller	32
4.2.2	Implementation of an actuator disc	33
4.3	Model setup	35
4.3.1	Turbulence modelling	35
4.3.2	Mesh creation	36
4.3.3	Boundary conditions and initial conditions	39
4.3.4	Numerical process	41
4.4	Model calibration	42
4.4.1	Flow generation in a simple 1D simulation	43
4.4.2	Comparison of the shape functions in a duct	44
4.4.3	Coefficient calibration to the efflux	45
4.4.4	Diffusion of the water jet	48
4.4.5	Velocities at a slope compared to theory	48
4.4.6	Velocities at a slope compared to measurements	49
4.5	Conclusion - Using the model	53
5	SENSITIVITY ANALYSIS	55
5.1	Changing the actuator disc	55
5.1.1	Thickness disc	55
5.1.2	Location actuator disc	56
5.1.3	Adding radial velocities at the actuator disc	56
5.1.4	Influence of the tangential component	57
5.2	Changes to the mesh	58
5.2.1	Mesh of the propeller duct	58
5.2.2	Mesh at the slope	59
5.2.3	Confined or unconfined jet	60
5.3	Wall function roughness	61
5.4	Turbulence models	62
5.5	Efflux velocities	65
5.6	Conclusion - Sensitivity of the model	65
6	RESULTS FOR CHANGING GEOMETRY	67
6.1	Distance to the slope	67
6.2	Angle of the slope	68
6.3	Addition of piles	69
6.4	Quay walls	72

6.5	Oblique walls	73
6.6	Full-size model	73
6.7	Conclusion of the results	74
7	CONCLUSIONS AND RECOMMENDATIONS	75
7.1	Conclusions	75
7.1.1	Preparations for the numerical model	75
7.1.2	Model set-up	75
7.1.3	Model results	76
7.2	Recommendations	77
	REFERENCES	78
	NOMENCLATURE	81
	ABBREVIATIONS	83
	LIST OF TABLES	83
	LIST OF FIGURES	84
	APPENDICES	86
	APPENDIX A TURBULENCE MODELLING	87
A.1	Summary of turbulence models	87
A.2	Equations of the realisable k-epsilon model	88
	APPENDIX B EQUATIONS OF HOUGH AND ORDWAY	89
	APPENDIX C SCALE MODEL RESEARCHES	90
C.1	Open quay structures: Van Doorn	90
C.1.1	Correction to the parallel velocities	90
C.1.2	Model and scenario dimensions	91
C.1.3	Measurement locations	92
C.1.4	Maximum measured velocities	94

C.2	Closed quay wall: Van Blaaderen.....	96
C.2.1	Model dimensions.....	96
C.2.2	Measurement locations.....	96
APPENDIX D	CONSIDERED MODELLING SOFTWARE	97
APPENDIX E	BODY FORCE IMPLEMENTATION IN THE SOLVER	99
APPENDIX F	DISCRETISATION SETTINGS	105
APPENDIX G	IMPLEMENTATION AS BOUNDARY CONDITION	107
APPENDIX H	FULL CALIBRATION FIGURES	108
APPENDIX I	RESULTS AT PILES INCLUDING TURBULENCE	113

1 INTRODUCTION

During mooring operations the flow velocities induced by a vessel's (bow) thruster generates a highly turbulent water jet resulting in erosion of the bed. Current design approaches have large uncertainties for the calculation of the flow velocities and are not designed for the calculation at specific structures at quay walls. In this thesis report a numerical model is applied to better predict the flow velocities of a bow thruster at an open quay structure.

This chapter introduces the use of bow thrusters and the resulting problem at the quay structures. To solve this problem the objective and research questions are formulated and a methodology is presented which is used as guideline in the research towards the conclusions of this report.

1.1 EROSION INDUCED BY BOW THRUSTERS

The propulsion of vessels is predominantly done with thrusters located at the stern of a vessel. Steering of a vessel needs a forward or a backward velocity, which might not be possible at quay structures where other vessels are moored in the close proximity of the vessel. In many situations the mooring procedure will be aided with tugboats, but this can be expensive and time consuming.

To improve the manoeuvrability of vessels extra thrusters are installed which generate a transverse water jet (left of Figure 1-1). When located at the bow of a vessel they are named bow thruster. These days most inland and ocean-going vessels have a bow thruster installed. This allows for manoeuvres for the (de-)berthing to be performed as shown in the right Figure 1-1 where minimal free space at the quay is available.

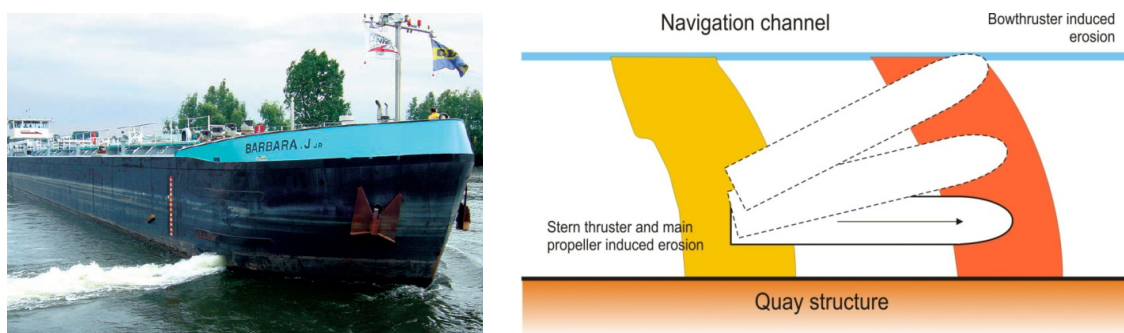


Figure 1-1: (left) Working principle of a bow thruster; (right) De-berthing manoeuvre at a quay structure with a bow thruster [PIANC MarCom, 2013]

Near quay walls the jet generated by the bow thruster during the (de-)berthing process introduces severe velocities and turbulence at the bottom and the slope which can result in erosion of the (sloped) waterbed.

At a structure the flow is obstructed and redirected in all directions generating complicated flow patterns. In Figure 1-2 the two-dimensional deflection at a closed quay wall is shown where, besides flow velocities in lateral directions, a strong jet flow is deflected towards the bottom. When the forces of the jet load exceed the strength of the bed, the soil will erode and lead to the appearance of scour holes as shown at the right of Figure 1-2. It is obvious that the water jet of the main propeller and of the bow thruster leads to scour holes at different locations, as labelled in the figure. While the main propellers lead to scour in the main channel, the bow thrusters will erode the soil at the foot of the quay structure. This results in a reduction of the passive ground pressure and can eventually lead to a collapse of the quay wall [PIANC MarCom, 2013].

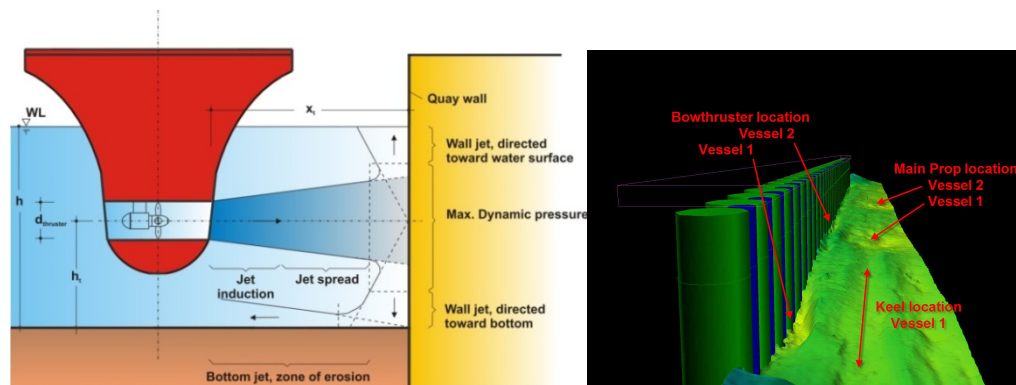


Figure 1-2: (left) Jet spreading at a closed quay wall; (right) Scour holes at a quay wall [PIANC MarCom, 2013]

Therefore it is of importance that the bed near a quay structure has sufficient strength by either the soil parameters or by placing a bed protection. To dimension this protection the flow velocities are approximated with basic design formulae. In the simple situations these give good approximations and are combined with extra safety magnification factors for a secure design. At more complicated situations like an open quay structure (Figure 1-3) where the effect of both piles and a sloped bed need to be taken into account, the formulae have a large inaccuracy and extra safety is included in the calculations, possibly leading to an oversized or still undersized bed protection.

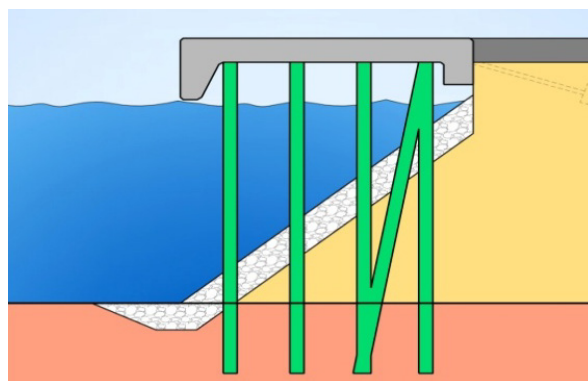


Figure 1-3: Open quay structure supported by piles over a sloped bed [PIANC MarCom, 2013]

A more accurate prediction can be made with the use of a numerical model. A properly calibrated model will show more insight in the flow patterns and the effects of different structures on the velocities.

1.2 OBJECTIVE AND RESEARCH QUESTIONS

In this thesis project a numerical model is built that generates a correct reproduction of the flow velocities induced by a bow thruster. The objective is formulated as follows:

- > *The set-up of a 3D numerical model to analyse the flow velocity induced by bow thrusters at open quay structures for a non-erodible bed.*

To arrive at this objective the following partial research questions are answered:

- > *What are the physics of bow thrusters?*
- > *What measurement data is available and how accurate is this data?*
- > *Which flow patterns can be noticed in the data and should be incorporated in the model?*
- > *What is the best numerical software package for the development of a bow thruster model?*
- > *Does the developed numerical model agree with the scale model measurements?*
- > *How well does the model respond to sensitivity analyses?*
- > *What changes can be noticed in the model for different open quay structure geometry?*

Ultimately leading to the answer of the final research question:

- > *What are the normative flow velocities of a bow thruster at a predefined open quay structure?*

1.3 RESEARCH METHODOLOGY

For the answering of these questions the research methodology described below is used. This is also used as the outline for this report and therefore the different paragraphs correspond to the chapters of this thesis report.

At first the theoretical background of a bow thruster is discussed in *Chapter 2*. A literature study is done regarding the physics of a bow thruster. This also includes a method to approximate the velocities generated by a propeller by approaching the geometry of the propeller blades with a circulation distribution. This chapter concludes with the simplified approach as applied in design practice.

For the calibration of the numerical model no new measurements in a physical (scale) model were done, but data of previous studies is used and analysed. A decision is made for which measurements the main calibration can be made based on the amount of available data, the relevance and the accuracy in *Chapter 3*. Besides looking at these previous scale model researches, an evaluation is also made of the earlier numerical models that were built.

A decision is made for a numerical software package in *Chapter 4*. Within this package the bow thruster propeller is implemented as a simplified actuator disc and calibrated and compared to the chosen measurements and the design practise theory.

For this model a sensitivity analysis is done for a number of numerical parameters in *Chapter 5*. The different results give an indication of the stability of the bow thruster simplification and show possible improvements to some parameters of the model configuration.

In *Chapter 6* velocities are computed for different geometries of quay structures and compared to the theoretical velocities. This results in insight in the deflection of the jet of a bow thruster at a sloped open quay structured with piles and other structures.

Finally, in *Chapter 7* an overview of the conclusions and the recommendation of this research is presented.

2 THEORETICAL BACKGROUND

In this chapter some of the theory for a propeller used in a bow thruster is given, starting with a description of a typical propeller and bow thruster in *Chapter 2.1*. For a simple and optimal propeller, the expected velocities are calculated as a function of the radius for a steady situation. This approach, in which the circulation distribution is converted to velocities, is applied for both an open and a ducted propeller in *Chapter 2.2*.

In design practise the calculation of the flow field is done with a more simple approach. In *Chapter 2.3*, approximations based on the momentum theory are used to calculate the convection and diffusion of the water jet generated by the propeller. As one of the suggested standard design approaches, the 'Dutch method' is presented.

The goal of calculating the velocities is to ultimately determine the induced erosion. Therefore the link to an approach to calculate the erosion is given in *Chapter 2.4*. This shows which flow parameters need to be correctly computed to be able to estimate the erosion and the resulting scour hole.

2.1 PHYSICS OF A BOW THRUSTER

As shown in Chapter 1.1 a bow thruster is defined as a ducted thruster constructed within the hull of the ship to exert a transverse force for mooring operations. It is constructed as close to the bow of ship as possible to be able to exert the highest moment of force on the ship.

One can distinguish two types of bow thrusters. The first type is the common type in which a duct is fixed in transversal direction and hence also named transverse thruster (Figure 2-1). Within the duct a propeller is able to rotate in either clockwise or anti-clockwise direction to generate a water jet at port or starboard. These thrusters lose their efficiency at sailing speeds above 2 knots [PIANC MarCom, 2013].

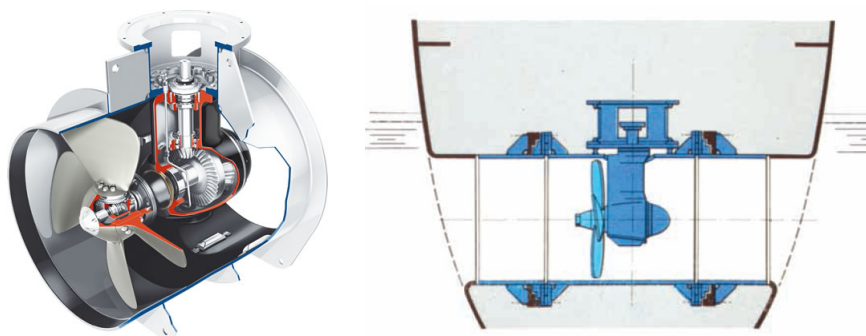


Figure 2-1: 2D and 3D view of a transverse (bow) thruster [Schottel, sd]

The second type of bow thruster is called a pump jet thruster and is originally designed for inland navigation, where conventional propulsion systems are not always applicable due to the shallow water depth. In contrast to a transverse thruster, a pump jet thruster has the inflow located in the keel of the ship and an outflow that can freely rotate in all directions (Figure 2-2). This makes it possible to use it for both propulsion and enhanced manoeuvrability purposes.

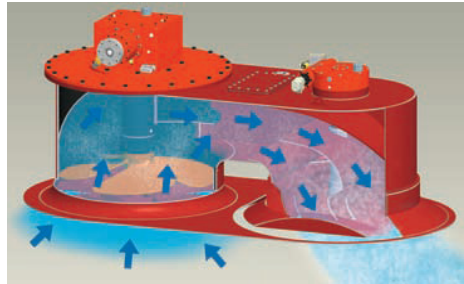


Figure 2-2: Principle of a pump jet thruster ¹

Although the outflow of both types of thruster might show large similarities, this thesis solely includes the transverse thruster, which is simply called bow thruster afterwards.

As it is used in both inland and ocean operating vessels a large variability in dimensions and quantity exists. Figure 2-3 shows a small bow thruster in a cruiser yacht at the left with a diameter of only a few decimetre and several kW of power. The right figure shows the large cruise vessel *Oasis of the Seas* which uses four large bow thrusters of several metres diameter with a power of 5.5 MW (7,800 hp) each.



Figure 2-3: (left) A bow thruster in a small cruiser yacht; (right) Four bow thrusters in a very large cruise vessel. Both marked with a yellow circle

The propeller located within the duct works with the same principle as any normal propeller. The propeller blades are connected to the hub and rotate around the center. As it rotates, it pushes the water away on one side of the blade, while at the other side the water rushes in to fill the space left by the moving blade. This results in a difference in pressure between, respectively, the pressure side and the suction side of the propeller blade. This pressure difference cause the water to be drawn into the propeller and to be accelerated into a water jet. The acceleration is both applied in axial direction (x) as well as in tangential direction (θ). These radial axis definitions, along with the cartesian axis system which is used in this thesis,

¹ Source: Veth motoren, www.veth-motoren.com

are shown in Figure 2-4. The tangential direction can be derived using the right hand rule to the x-axis.

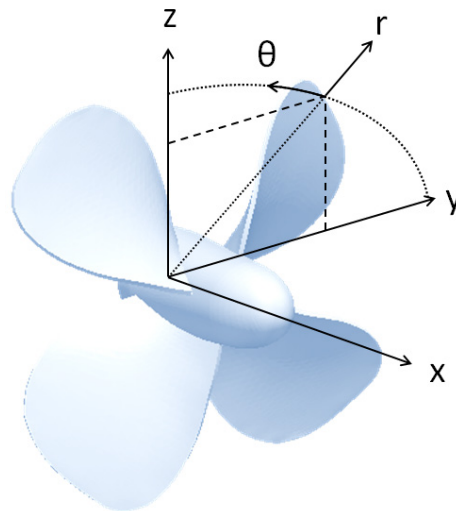


Figure 2-4: Positive axis definitions referred to as axial (x), lateral (y), vertical (z), radial (r) and tangential (θ) direction. A positive value of x corresponds with the governing flow direction.

When too much power is transmitted through the screw and the propeller is operating at high speed or under heavy load the pressure at the suction side can drop below the vapor pressure of water, resulting in the formation of vapor bubbles. This effect, called cavitation, results in a loss of thrust and damage to the propeller, but will also generate high local velocities. Figure 2-5 shows that the cavitation in the flow is generated at two specific locations, viz the hub and the tip of the propeller blades. It can also be seen that the cavitation hardly reduces in axial direction and that the diameter of the water jet only slightly contracts.

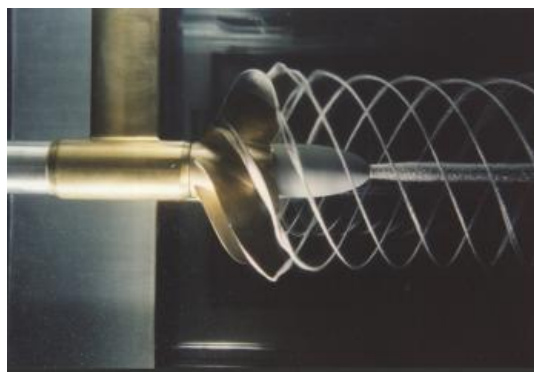


Figure 2-5: Cavitation at a propeller ²

Also at lower rotational velocities, when no cavitation occurs, these locations show a high vorticity (the local spinning motion of a fluid) resulting in a high local velocities.

Bow thrusters show a lot of similarities with the propeller design of the ducted propeller, although the length of the duct is much longer. In both cases often a Kaplan type propeller is applied. These have an increased thrust by cutting off the end of the propeller blades. Figure

² Source: http://www.bbc.co.uk/iplayer/episode/b00rmrln/Richard_Hammonds_Invisible_Worlds_Speed_Limits/

2-6 shows both a traditional shaped propeller in a duct and a ducted propeller with Kaplan shaped blades.



Figure 2-6: Ducted propellers with (left) traditional propeller blades and (right) Kaplan type propeller blades ³

2.2 APPROXIMATING THE PROPELLER WITH AN ACTUATOR DISC

A method to calculate the thrust and torque generated by a propeller, is the determining of the vortices generated by the propeller. A vortex is a region where the flow is mostly spinning about an imaginary axis. At a propeller two type of vortices can be recognised, the bound vortices at the propeller blade and the trailing (or free) vortices in the stream generated by the propeller. Figure 2-7 shows both schematically.

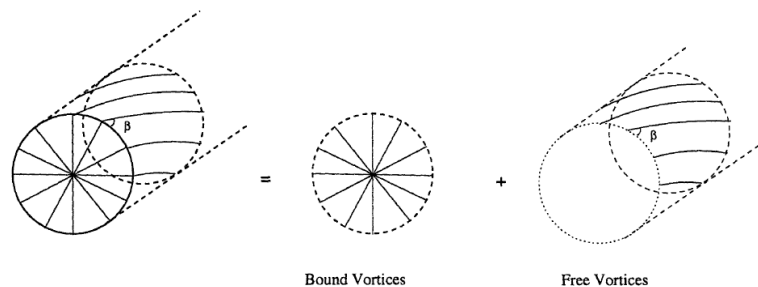


Figure 2-7: Decomposition in bound and free vortices of Hough, et al. [Buchoux, 1995]

The total strength of the vortices can be formulated with the circulation (Γ), which is defined as the line integral about a closed curve of the velocity field. In the approach done below, the geometry of the propeller is replaced by a radial distribution of the circulation. By using methods based on the law of Biot-Savart it is possible to approximate the induced velocities from the distribution of the circulation.

2.2.1 FREE PROPELLERS WITH THE GOLDSTEIN OPTIMUM

Goldstein developed an approach to calculate the circulation distribution along a propeller blade based on the vortex theory for propellers as an addition to the approximations previously done by Prandtl [Goldstein, 1929]. As minimum loss of energy in the slipstreams is one of the basic principles of the theory, it is also referred to as the *Goldstein optimum*. For the Goldstein optimum the distribution of the circulation can be numerically calculated as a function of the given thrust, radius and number of propeller blades.

³ Source: http://www.solarnavigator.net/kort_nozzle.htm

As the set of equations given by Goldstein was comprehensive and had high computational costs at that time, Hough, et al. found the need to develop simple, yet accurate formulae of a free propeller that could be applied within aerodynamic and naval scopes [Hough & Ordway, 1964]. In their approach they decomposed the vortices in the bound and free vortices of Figure 2-7. They made a Fourier analysis of the resulting set of equations and took the zeroth harmonic, also called the steady component, to make further simplification possible. These steady components made it possible to calculate the velocities in axial (U_x), tangential (U_θ) and radial (U_r) direction as a function of the circulation ($\Gamma(r)$), the number of propeller blades (Z), the propellers angular velocity (Ω) and the free velocity at a far distance (U). This full set of equations is given in Appendix B.

The velocities in radial direction show a symmetric logarithmic profile in axial direction and are zero at the location of the propeller. For the axial and tangential velocities at the location of the propeller ($r \leq R$ and $x = 0$) the functions of Appendix B can be significantly reduced to equation (2.1) and (2.2).

$$U_x = \frac{Z\Omega}{4\pi U} \Gamma(r) \quad (2.1)$$

$$U_\theta = \frac{Z}{4\pi r} \Gamma(r) \quad (2.2)$$

To approximate the Goldstein optimum a shape function was created which proofed to have a good fit for representative distributions of the circulation distribution, shown in Figure 2-8. The constant (A) is a function of, amongst others, the thrust coefficient and the propeller advance ratio ($J = U/\Omega R$).

$$\frac{\Gamma}{UR} = A \frac{r}{R} \sqrt{1 - \frac{r}{R}} \quad (2.3)$$

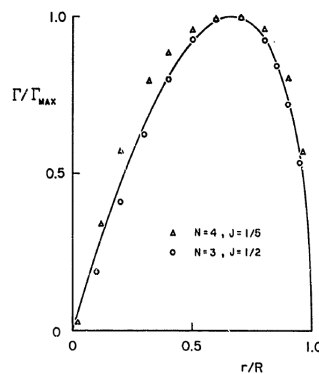


Figure 2-8: Comparison of the representative circulation with two examples of the Goldstein optimum [Hough & Ordway, 1964]

The induced velocities at the propeller can also be written as a momentum, or body force, function. This link is shown in Equation (2.4) and (2.5). As the implementation later on requires the disc to be three dimensional, the area is multiplied with the thickness (Δ).

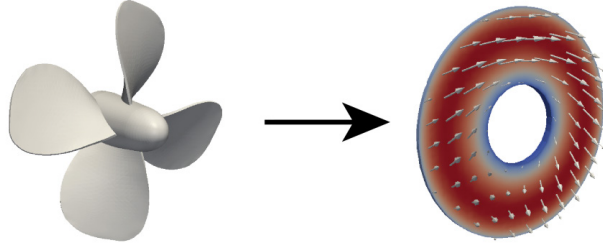


Figure 2-9: The conversion of a propeller to an actuator disc

$$T = K_T \rho n^2 D_P^4 = \rho \int u_x^2 dA = \Delta \int f_x dA \quad (2.4)$$

$$Q = K_Q \rho n^2 D_P^5 = \rho \int u_\theta^2 r dA = \Delta \int r \cdot f_\theta dA \quad (2.5)$$

With the circulation shape function of Hough & Ordway in Equation (2.3) and the derived velocities in Equation (2.1) and (2.2), the body force distribution functions are created by Stern, et al. with Equation (2.4) and (2.5) and are shown in Equation (2.6) and (2.7) [Stern, et al., 1988]. The derivation of Hough & Ordway did not include the hub of the propeller, but the presence of the hub was included by Stern, et al. by adjusting the radius to a relative radius including this hub. In Figure 2-10 their computed distribution of the axial and tangential velocities is shown.

$$f_x = A_x r^* \sqrt{1 - r^*} \quad (2.6)$$

$$f_\theta = A_\theta \frac{r^* \sqrt{1 - r^*}}{r^* \left(1 - \frac{R_h}{R}\right) + \frac{R_h}{R}} \quad (2.7)$$

In which:

A_x = axial coefficient

A_θ = tangential coefficient

$r^* = \frac{r - R_h}{R - R_h}$ = relative radius

r = distance from center point

R_h = radius to the root of the propeller (or hub radius)

R = radius to the tip of the propeller

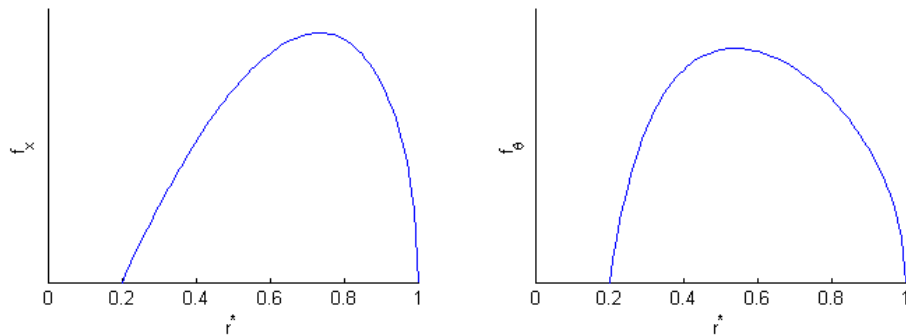


Figure 2-10: Hough and Ordway's force distribution in axial directions (left) and tangential directions (right)

The coefficients in those equations were originally defined as a function of the thrust coefficient (K_T) and the torque coefficient (K_Q) with the relation to the thrust and torque as shown in Equation (2.4) and (2.5). They can be rewritten as a function of the total thrust (T) and torque (Q) by using the following definition [Wolfgang, 2011].

$$T = \Delta \int_A f_x dA = 2\pi\Delta \int_{R_h}^R f_x r dr \quad (2.8)$$

$$Q = \Delta \int_A r \cdot f_\theta dA = 2\pi\Delta \int_{R_h}^R f_\theta r^2 dr \quad (2.9)$$

This resulted in the definition of the axial and tangential coefficient as shown below, as corrected from [Svenning, 2010].

$$A_x = \frac{105}{8} \frac{T}{\pi\Delta(4R + 3R_h)(R - R_h)} \quad (2.10)$$

$$A_\theta = \frac{105}{8} \frac{Q}{\pi\Delta R(4R + 3R_h)(R - R_h)} \quad (2.11)$$

2.2.2 DUCTED PROPELLERS

For a bow thruster not the comparison to an open propeller, but to a ducted propeller should be made. This was already shown in Chapter 2.1, as the blades of the propeller are shaped different for ducted propellers, but also because the presence of a duct changes the flow and results in a different shape of the circulation distribution. In Figure 2-11 the distribution is shown for several gap widths [Coney, 1989] [Stubblefield, 2008].

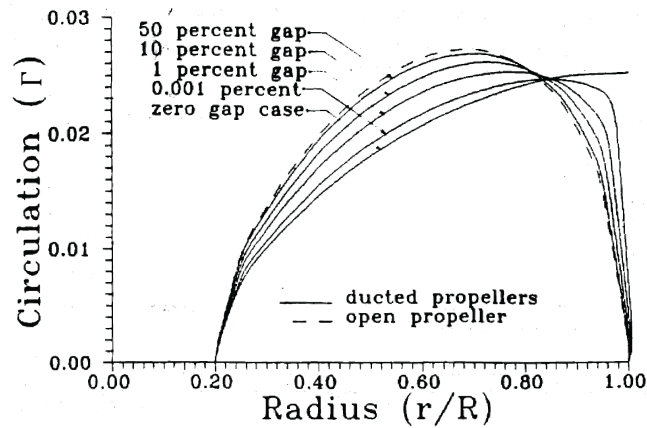


Figure 2-11: Circulation distribution for a ducted propeller for different gaps between propeller blade and duct [Coney, 1989]

To be able to measure the maximum influence of a duct due to the body force functions, the zero gap case is further used to calculate its derive its body force functions. A formula fit is created in the form of Equation 2.12, and fitted to the zero gap case in Figure 2-12.

$$\Gamma = c_1 + c_2 \cdot e^{c_3 \cdot r} \quad (2.12)$$

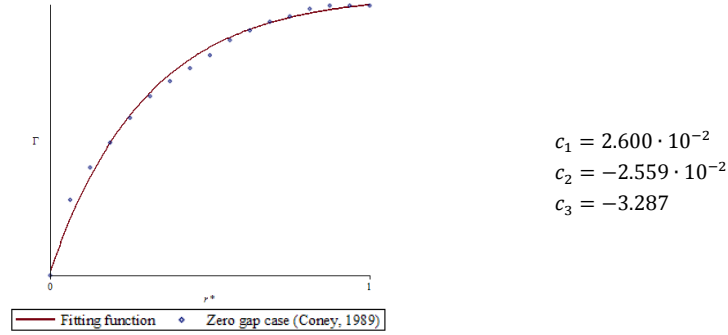


Figure 2-12: Function fitting to the zero-gap ducted propeller. The dots are the measured points from Figure 2-11

The velocities generated by this momentum are calculated in two approaches. At first an approach is used in which the propeller lifting line theory is applied. This is later on compared to a similar approach as done by Stern, et al [1988] as presented in Chapter 2.2.1.

For the first approach the velocities (or momentum) generated by this propeller are calculated with the propeller lifting line theory [Coney, 1989] [Stubblefield, 2008] [Epps, 2010]. The propeller lifting line theory is an application of the Biot-Savart law where the propeller is presented as a set of Z straight radial lines, one for each blade of the propeller, with an identical circulation distribution for each line. It assumes the steady time-averaged propeller forces, making the flow velocity vary only radially and not circumferentially.

By locally applying the Kutta-Joukowski's law the lift forces are calculated and a function for the total velocity V^* and the angle with respect to the plane of rotation β_i as shown in Figure 2-7. In these equations u_x^* and u_θ^* are the axial and tangential component of the induced velocities and combine with the effective inflow components V_x and V_θ and the propeller's rotation ωr .

$$V^*(r) = \sqrt{[V_x(r) + u_x^*(r)]^2 + [\omega r + V_\theta(r) + u_\theta^*(r)]^2} \quad (2.13)$$

$$\beta_i(r) = \tan^{-1} \left[\frac{(V_x(r) + u_x^*(r))}{\omega r + V_\theta(r) + u_\theta^*(r)} \right]$$

The total forces on the fluid can now be expressed in the propeller thrust and torque. The formula also includes the viscous drag force which is added in a direction perpendicular to V^* .

$$T = \rho Z \left[\int_{r_h}^R V^* \Gamma \cos \beta_i - \frac{1}{2} (V^*)^2 c_{Dv} \sin \beta_i \right] dr$$

$$Q = \rho Z \left[\int_{r_h}^R V^* \Gamma \sin \beta_i - \frac{1}{2} (V^*)^2 c_{Dv} \cos \beta_i \right] r dr \quad (2.14)$$

For the solving of these equations to a body force function a couple of assumptions are done. The contribution of the drag is neglected as it is assumed to only have a minor influence and as the drag coefficient (C_D) is not known. It is also assumed that the total axial velocity is

$u_x = u_x^* + V_x$ and the total tangential velocity is $u_\theta = u_\theta^* + V_\theta$ as the propeller is at a fixed position, making the induced velocity equal to the inflow velocity. Now using the definition of thrust and torque, the equations can be solved for the velocity components for a given circulation distribution.

$$T = \rho \int u_x^2 dA = 2\pi\rho \int u_x^2 \cdot r \, dr = \rho Z \int \Gamma \cdot (\omega r + u_\theta) \, dr \quad (2.15)$$

$$Q = \rho \int u_\theta^2 r \, dA = 2\pi\rho \int u_\theta^2 \cdot r^2 \, dr = \rho Z \int \Gamma \cdot u_x \cdot r \, dr \quad (2.16)$$

Of the four solutions to these of equations only one solution is real and not asymptotic at the origin for both the axial and tangential velocity and shown in Figure 2-13.

As an additional calculation to confirm the shape of these functions, the method used by Stern, et al. is applied [Stern, et al., 1988], which is simply a change in coefficient for the axial velocity and an additional division by the radius for the tangential component. This resulted in a similar distribution with simpler formulae, shown in Equation (2.17) and (2.18).

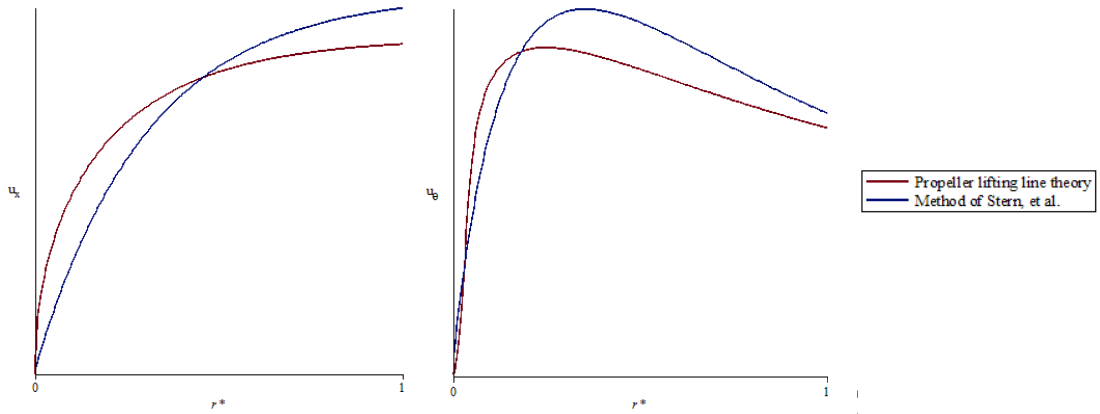


Figure 2-13: Velocity distribution for a ducted propeller in axial (left) and tangential (right) directions for the propeller lifting line theory and the method applied by Stern, et al [1988].

$$f_x = A_x \cdot (c_1 + c_2 \cdot e^{c_3 r}) \quad (2.17)$$

$$f_\theta = A_\theta \cdot \frac{c_1 + c_2 \cdot e^{c_3 r}}{r^* \left(1 - \frac{R_h}{R}\right) + \frac{R_h}{R}} \quad (2.18)$$

For these functions the axial and tangential coefficient are determined by using Equation 2.15 and 2.16. Instead of an analytical derivation, the equations are numerically integrated and can be solved to the coefficients for a given thrust and torque.

2.3 FLOW VELOCITIES IN DESIGN APPROACHES

For civil engineering structures the design velocities are not calculated with the circulation method of Chapter 2.2. Instead a simplified representation of the flow field is made with formulae based on both the momentum theory and the examining of measurements.

In a previous research regarding bow thrusters a full overview was already generated and a comparison was made between the approaches [Van Doorn, 2012]. It was concluded that the combination of formulae which is referred to as the ‘Dutch approach’ show good results and was used for comparison to his scale model measurements. Along with the ‘German approach’ this methodology is included in the guideline for scour at berthing structures by thrusters [PIANC MarCom, 2013]. For these reasons it is chosen to only elaborate the Dutch approach. Many of the theorems and assumptions that are elaborated below, are done in a similar way in the other methods but concluded small changes of the coefficients.

For the calculation of the bed protection several steps are taken and elaborated below. At first the velocity just at the outflow side of the propeller is calculated, which is referred to as the efflux velocity. This velocity is used to calculate the flow field in axial direction of the propeller and for each location in radial direction the flow field can be computed. The last phase contains the calculation of the maximum bed load in which the formula is extended to include slopes, walls or piles.

2.3.1 EFFLUX VELOCITY

The research of the momentum generated by a propeller is based on the 19th century axial momentum theory of Froude and the research of Alberston, et al. on the diffusion of a submerged jet [Albertson, et al., 1948]. Blaauw and van de Kaa developed a formula to predict the velocity generated by the propeller in a research into the erosion of propellers [Blaauw & van de Kaa, 1978].

Several assumption were done to construct a simple formula. It is assumed that the propeller has an infinite number of blades, rotating with an infinite velocity and the generated load is constant over the radius with no propeller hub. Furthermore it assumed that the thickness of the propeller in axial direction is negligible, is submerged in an ideal fluid without disturbances and the energy generated by the propeller is only supplied in axial direction. Figure 2-14 shows this situation with an already present flow velocity (U_A) and the velocity gain by the ducted propeller at the propeller (U_1) and at a far distance (U_2).

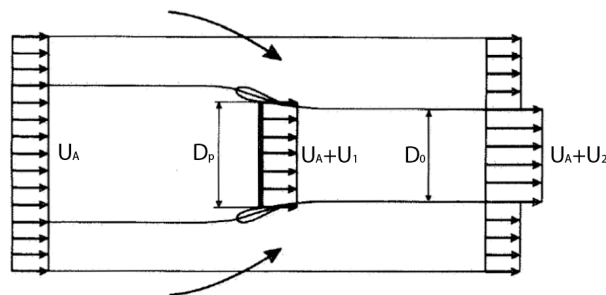


Figure 2-14: Control volume for momentum theory propellers. Based on [Blaauw & van de Kaa, 1978]

Using Bernoulli's principle, stating that an increase in the fluid speed results in a decrease of the potential energy (or the pressure) and the conservation of mass, a formula is generated for the bollard pull condition ($U_A = 0$) [Blaauw & van de Kaa, 1978].

$$U_0 = 1.60 \cdot n \cdot D_0 \cdot \sqrt{K_t} \quad (2.19)$$

In this formula the efflux velocity (U_0) is equal to the far distant velocity (U_2) and described as a function of the thrust coefficient (K_t), the rotational speed of the propeller (n) and the diameter of the contracted water jet (D_0). The contraction of the water jet (shown in Figure 2-5) depends on the type of propeller and is usually $D_0 = 0.71 \cdot D_p$ for non-ducted propellers and $D_0 = D_p$ for ducted propellers, where D_p is the diameter of the propeller.

However, the thrust coefficient is not always known and has to be assumed based on the design-diagrams. It can be approximated as a function of, amongst others, the power of the engine (P) and the diameter of the propeller (with the research of Schneiders and Pronk), but it can also be incorporated in the above formula as done by Verheij [Verheij, 1985].

$$U_0 = 1.15 \cdot \left(\frac{P}{\rho D_0^2} \right)^{\frac{1}{3}} \quad (2.20)$$

For bow thrusters it is preferred to use the slightly larger diameter of the thruster's duct instead of the propeller diameter as this is more often known. Fortunately the energy losses in the duct can be assumed to be equal to the velocity decrease by this increased diameter resulting in the assumption $D_0 \approx D_{thruster}$.

2.3.2 FLOW FIELD OF AN UNCONFINED JET

The velocity field in axial direction this field is divided in two zones (Figure 2-15). Directly behind the propeller the zone of flow establishment is defined in which the flow is still developing. At the transition point, located at 2.8 times the propeller diameter, this changes in the zone of established flow. The zone of flow establishment has no decay in the maximum velocity and has a change in location of the highest velocities from the propeller tip to the centre. In the zone of established flow the highest velocity is at the centre of the flow and reduces in axial direction due to radial diffusion.

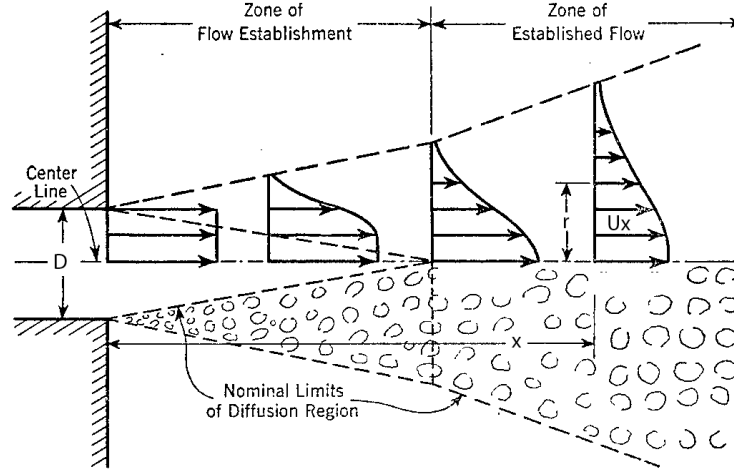


Figure 2-15: The zone of flow establishment and zone of established flow by [Albertson, et al., 1948]

This theory, which was introduced by Albertson, et al. for submerged jets, was applied to propellers by Blaauw & Van de Kaa. The formula consists of a part representing the decay of the maximum velocity in the axial direction and a part representing the dispersion in radial direction. Further assumptions are a dynamically similar diffusion process under all condition with a normal distribution in radial direction [Blaauw & van de Kaa, 1978].

The resulting flow field (U) caused by a single propeller is a function of the axial distance to the outflow (x), the radial distance to the axis (r), the efflux velocity (U_0) and the jet diameter (D_0). The constants (A , a and b) are determined by experiments. For the Dutch approach these constants are defined as $A=2.8$, $a=1$ and $b=15.4$.

$$U(x, r) = A \cdot U_0 \cdot \left(\frac{D_0}{x}\right)^a \cdot \exp\left(-\frac{b \cdot r^2}{x^2}\right) \quad (2.21)$$

For determining the velocities at a horizontal bed with no nearby obstructions this formula can be used to calculate the maximum velocities at the bottom. For this, the radius (r) is equal to the height of the propeller above the horizontal bed (h_{pb}). The maximum velocity at this radius is, for the Dutch method, located at a distance $x_{max} = 5.6 \cdot h_{pb}$. The maximum velocity at this location caused by a single propeller can be calculated with Equation (2.22).

$$U_{b,max} = 0.306 \cdot \frac{U_0 \cdot D_0}{h_{pb}} \quad (2.22)$$

2.3.3 VELOCITIES AT A SLOPE

The velocities at a slope can as a first approximation be calculated with Equation (2.21) of Blaauw and Van de Kaa for an unconfined jet. This gives an approximation of the actual velocities, as the axial propagation and the radial spread of the jet are restricted by the slope surface, causing the jet to be not unconfined at the slope. For the radial coordinate r along the slope surface and the adjacent horizontal bottom the following equations hold:

$$r = \sqrt{y^2 + z^2} \quad (2.23)$$

with

$$z = \begin{cases} h_{pb}, & x \leq x_{toe} \\ z_{slope}(x), & x_{toe} \leq x \leq L \\ 0, & x \geq L \end{cases} \quad (2.24)$$

In this equation h_{pb} is the height of the jet axis above the horizontal bottom adjacent to the slope, x_{toe} is the x-coordinate of the toe of the slope, $z_{slope}(x)$ is the z-coordinate of the slope at x and L the horizontal distance between the outflow of the thruster and the intersection of the jet axis with the slope.

The motivation for the value of $z = 0$ at higher location on the slope is that the jet does not pass through the slope, but instead flows upward the slope for $x > L$. In this region the axial propagation distance should theoretically be measured along the slope (Equation 2.25). However, this refinement can for simplicity be omitted, considering the fact that the equation for an unconfined jet gives only a rough approximation of the real jet behaviour at $x > L$.

$$x = L + \sqrt{(x - L)^2 + z_{slope}^2} \quad (2.25)$$

Based on the above assumptions and formula of Blaauw and van de Kaa for an unconfined jet, Blokland has derived a formula for the maximum flow velocity on a slope with angle β . In this equation a correction factor f is included to bring into account the confinement of the jet by the slope surface. Blokland determined values of f with the results of scale model measurements [PIANC MarCom, 2013].

$$U_{slope,max} = f \cdot A \cdot \left(\frac{D_0}{L} \cdot \frac{L}{x_{U,max}} \right)^a \cdot U_0 \cdot \exp \left(-b \cdot \left(\frac{\frac{L}{x_{U,max}} - 1}{\cot(\beta)} \right)^2 \right) \quad (2.26)$$

The location ($x_{U,max}$) where this maximum velocity according to the formula for an unconfined jet theoretically occurs can be calculated with the following formulae.

$$x_{U,max} = K \cdot \left(\sqrt{1 + \frac{2}{K}} - 1 \right) \cdot L \quad (2.27)$$

$$K = \frac{b}{a \cdot \cot^2(\beta)} \quad (2.28)$$

Scale model measurements done by Van Doorn made it possible to determine the correction factor for the maximum velocities for some cases [Van Doorn, 2012]. Blokland derived the following values of the correction factor (f) [PIANC MarCom, 2013].

For a 1:2.5 slope with a smooth surface	f = 1.1
For a 1:1.5 slope with a rough surface	f = 1.25
For a 1:1.5 slope with a rough surface and piles	f = 1.4 - 1.5

The measurements also showed that the location of the maximum velocities ($x_{U,max}$) is close to the efflux than according to the theory.

The accuracy of these factors is doubtable, as a goniometric function to derive the parallel velocities was incorrect. The correction is shown in Appendix C.1.1.

2.3.4 VELOCITIES AT A VERTICAL WALL

As illustrated before in Figure 1-2, the water at a vertical quay wall jet is deflected in all direction. The PIANC guideline for the Dutch method is based on the research of Blaauw & Van de Kaa, Verheij and Blokland and gives the velocities at the horizontal bed as a function of the distance to the quay (L) and the height of the propeller above the bed (h_{pb}) [PIANC MarCom, 2013]. Equation (2.29) and (2.30) are plotted in Figure 2-16 and show that the velocity is assumed to be constant for short distances to the quay and decreases for larger distances.

$$\frac{L}{h_{pb}} < 1.8 \quad U_{b,max} = 1.0 U_0 \frac{D_0}{h_{pb}} \quad (2.29)$$

$$\frac{L}{h_{pb}} \geq 1.8 \quad U_{b,max} = 2.8 U_0 \frac{D_0}{L+h_{pb}} \quad (2.30)$$

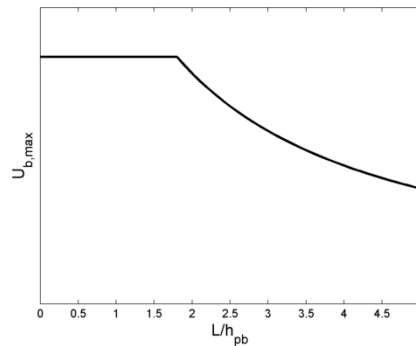


Figure 2-16: Maximum velocity at the bed as a function of the distance to the quay divided by the height of the propeller axis above the bed

2.3.5 VELOCITIES AT AN OBLIQUE WALL

An oblique wall is defined as a vertical wall at a small angle. For these quays a part of the water jet is still directed downward, but the magnitude reduces with increasing wall angle (left Figure 2-17).

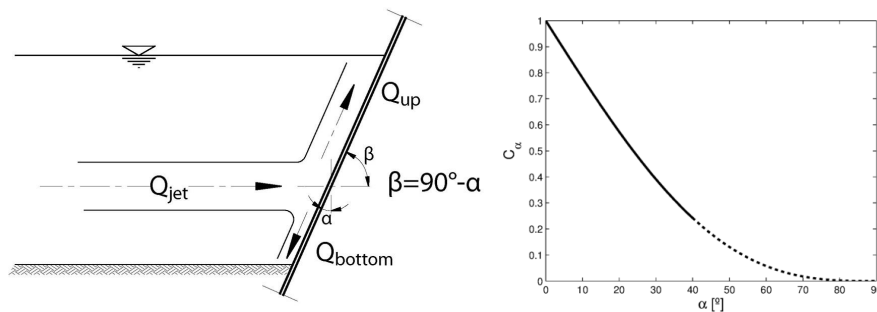


Figure 2-17: (left) Velocity distribution for an oblique wall; (right) Reduction factor for downward velocities at an oblique wall. The solid line indicates the scope of the factor

Römisch designed a method to determine the part of the jet which is directed to the bottom, shown in Equation (2.31). This method is a function of the angle α to a vertical wall and results in a reduction factor (C_α) to the velocities derived for a vertical wall in Equation (2.29) and (2.30). In the basic equations not the velocities, but the discharge (Q) ratio is used. It is assumed that the reduction factor C_α holds not only for the discharge ratio but also for the velocity ratio. This is confirmed by the calculations with the OPENFOAM model.

The formula was determined based on model tests up to an angle of 40 degree, which is also applied as maximum validity of the formula [PIANC MarCom, 2013]. At the right of Figure 2-17 this reduction is plotted, an angle α of 0° corresponds to a vertical wall and the maximum angle of 40° corresponds to a very steep slope of 1:0.83.

$$C_\alpha = \frac{Q_{bottom,\alpha}}{Q_{bottom,\alpha=0}} = \frac{1}{0.5} \cdot \left(\frac{90^\circ - \alpha}{180^\circ} - \frac{\sin(2 \cdot (90^\circ - \alpha))}{2 \cdot \pi} \right) \quad (2.31)$$

2.3.6 VELOCITIES AT PILES

When the structures gets more complicated the relation to the unconfined jet equation of Blaauw and van der Kaa reduces and the local velocities can increase significantly as a result of this structure. For these structures other physical relations need to be found or the local increase needs to be measured in a physical or numerical (scale) model.

An example of such structures is the open quay structure, where the water jet can be both influenced by a slope and by piles. An increase of the jet velocities on the slope was already given in Chapter 2.3.3, but at piles the flow is deflected and vortices result in higher flow velocities at the bed level. Figure 2-18 gives an indication of the complicated vortices within a steady flow.

As an approximation a simple rule of thumb can be applied, which says that in general the flow velocity adjacent to a pile will be twice the velocity of the approach velocity [Breusers, et al., 1977] [PIANC MarCom, 2013].

$$U_{pile} \approx 2 \cdot U_{approach} \quad (2.32)$$

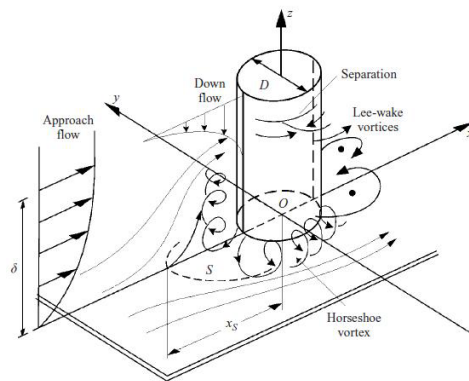


Figure 2-18: Characteristic features of the flow at a pile [Roulund, et al., 2005]

2.4 EROSION

At the bed or slope level, the water flow will pick-up sediment possibly leading to erosion when the deposition and entrainment fluxes are not in equilibrium. A qualitative description of the erosion process is given below to give an indication of the parameters that need to be correctly modelled. As the erosion itself is not included in this thesis the exact relations are not described.

Sedimentation is a function of the properties of the particles in the near-bed flow as the grain density, settling velocity and the near-bed concentration. The pick-up flux is depended on the turbulent velocities near the bed. Bursts of turbulence can pick up particles from the bed and inject them into the flow. For high erosion velocities, the near-bed concentration is significantly influenced by the injecting of the pick-upped particles and the turbulent eddies will also throws them back into the bed again.

The erosion depends on the Shields parameter, which describes the bed shear stress in a dimensionless form. Erosion can occur when the Shield number is higher than the critical Shield number. The sediment pick-up can be determined with the relation between the sediment pickup and the Shields parameter, which is known as a pick-up function. A well known example is the function by Van Rijn.

For high sedimentation velocities the dilatancy has an important effect on the erosion. This can be included in the existing pick-up functions by modifying the critical Shields parameter using the hydraulic gradient as an extra force on the grains [van Rhee, 2010].

2.5 CONCLUSION - INCREASE IN ACCURACY NECESSARY

It is concluded that the design approach in *Chapter 2.3* the velocities are calculated based on the theoretical momentum approach with additional empirical factors to include the difference of the analytical results to experimental measurements. For the velocities at a slope the velocities are amplified with a factor based on scale model measurements. Structures for which no model measurements have been carried out and for which consequently no factor exists will have a large inaccuracy for the velocities.

In a numerical model the flow patterns at different structures can be computed, but this requires the water jet to be accurately included. *Chapter 2.2* showed a derivation for both an open and a ducted propeller for inclusion within the numerical model. For this the geometry of the propeller was omitted and the propeller was simplified to a circulation distribution. With use of the lifting line theory, body force functions were derived and coupled to the thrust and torque of the flow.

It is of importance that the results of the model can be used for a prediction of the erosion of the bed. In *Chapter 2.4* it was concluded that both the velocities and the turbulence need to be accurately modelled in the numerical model.

3 MEASUREMENTS AND EARLIER NUMERICAL MODELS

Several researches are analysed to determine the amount and accuracy of the available measurement data, to be used as a basis for the design and calibration of the numerical model. The researches discussed below are the researches from which conclusions can be drawn for the numerical model. They are all done at the TU Delft and the measurements are freely available.

At first the measurements at sloped beds as done by Veldhoven [2002] and Schokking [2002] are presented in *Chapter 3.1*. With the addition of piles by Van Doorn [2012] this gives a good comparison to the flow at an open quay structure. At second in *Chapter 3.2* the measurements at closed quay walls of Van Blaaderen [2006] will be discussed.

In *Chapter 3.3* earlier numerical models, which were based on the measurements in the mentioned researches, are presented. The shortcomings of those models are analysed.

3.1 MEASUREMENTS AT OPEN QUAY STRUCTURES

In the past different researches were done comparing the velocity field generated by a pressure jet, an unducted (free) propeller and a ducted propeller on an open quay structure [Van Veldhoven, 2002] [Schokking, 2002]. As a bow thruster can be seen as a propeller in a long duct, these researches show features that need to be accounted for. The measurements were done with an electromagnetic flow meter (or EMS) in the set-up shown in Figure 3-1.

An EMS measures the voltage difference generated by a charge moving in a magnetic field. The sampling volume is not exactly known, but is relatively large compared to other measurement devices. Combined with the low measurement frequency of 10 Hz some of the turbulence and local velocities cannot be recorded. It is also only able to measure the velocities in two dimensions.

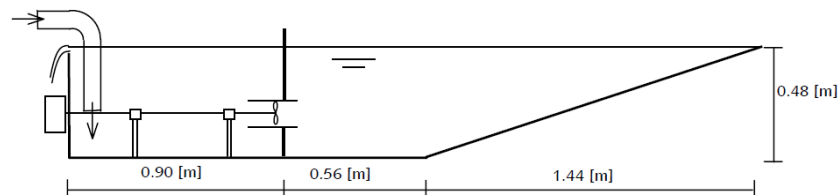


Figure 3-1: Set-up by Van Veldhoven and Schokking for scour at a slope

The outflow was measured in the vertical plane in front of bow thruster's efflux resulting in the velocities as shown in Figure 3-2. It can be seen that the radial diffusion is larger for a propeller than in the case of a pressure jet. This can be explained due to the induced tangen-

tial velocities and the higher turbulence intensity. The appearance of the propeller(hub) is clearly visible in the measurements up to a distance comparable to the zone of flow establishment as shown in Chapter 2.3.2. Comparing the free and the ducted propellers shows in the axial direction a stronger diffusion for the free propeller. This is probably caused by the smaller diameter of the efflux diameter for the free propeller.

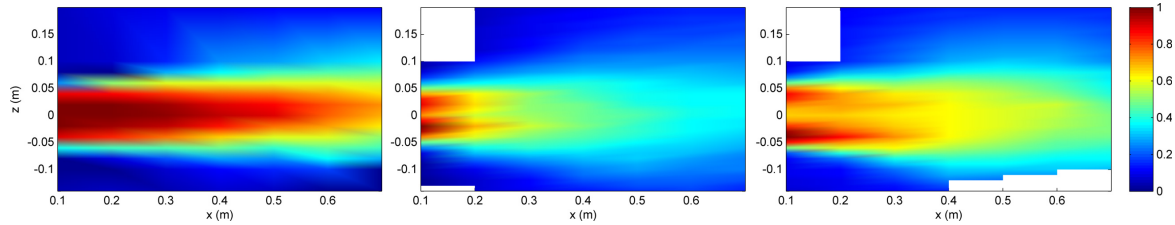


Figure 3-2: Relative velocities ($U/U_{\max}$) in axial direction. Measured for a pressure jet (left), free propeller (centre) and ducted propeller (right) by [Van Veldhoven, 2002] and [Schokking, 2002]

More extensive measurements were recently done by Van Doorn [Van Doorn, 2012]. A 1:25 scale model was built based of the container vessel Regina Maersk resulting in the dimensions shown in Appendix C.1. As the previous researches revealed that the implementation of the propeller was of significant influence on the resulting flow field, a real bow thruster was used (left of Figure 3-3) which is usually installed in small recreational vessels, like the yacht at the left of Figure 2-3. This thruster, built in a square vessel, was applied for ten different scenarios of open quay structures. Between the different scenarios the slope angle, water depth, roughness of the slope, distance between the thruster and the slope, the presence of piles and the alignment of these piles to the bow thruster were varied.

A narrow and wide basin was available, but previous measurements by the laboratory staff showed that it was not necessary to create the slope over the full width and a partial slope would be sufficient (right of Figure 3-3). The width of the basin was very limited, possibly influencing the run down due to circulation, but also limiting the slope. For most measurements a slope of 1:1.5 is used, but by reducing the water level an experiment with a slope of 1:2.5 was also conducted. In practise slopes vary between 1:1.2 and 1:4 [Van Doorn, 2012].

Measurements were done with an Acoustic Doppler Velocimeter (ADV) which is able to measure the flow velocities in a small sampling volume, with a high sampling rate (25 Hz) in three directions by measuring the velocities of added seeds in the flow.



Figure 3-3: (left) The Vetus bow thruster type 2512B used by Van Doorn; (right) Basin geometry of Scenario 10 of Van Doorn

The measurements results of Van Doorn showed that the velocities at the slope were higher than would have been expected at the location of the slope if the slope was not present. An amplification factor was derived and added to the formula as shown in chapter 2.3.3.

A large number of measurements were done, both close to the thruster and at the slope, making this ideal measurements for the validation of a numerical model. More details of the set-up of the different scenarios can be found in Appendix C.1. This appendix also shows corrections done to processing of the measured data to correct a goniometry mistake in the published results and the new maximum velocities including this correction.

From the measurements several characteristics of the flow can be derived, which are taken into account in the numerical model. For this analysis the basic formulae for the velocities are defined. The velocity exists of a mean part ($\langle U_i \rangle$) and a turbulent part (u_i).

$$\langle U_i \rangle = \frac{\sum U_i}{n} \quad (3.1)$$

$$u_i = \sigma = \sqrt{\frac{\sum (U_i - \langle U_i \rangle)^2}{n}} \quad (3.2)$$

Usually this turbulence intensity is converted to the turbulence kinetic energy (k), which is also used for the comparison to numerical model. In this process the vector information gets lost and a scalar value is left, but the turbulent energy is also modelled non-directional in steady state numerical models.

$$k = \frac{1}{2} (u_x^2 + u_y^2 + u_z^2) \quad (3.3)$$

3.1.1 EFFLUX VELOCITIES AND TURBULENCE

Van Doorn measured the velocities in three direction at a distance of $x = 95$ mm from the efflux. In Figure 3-4 the velocities are shown for all three directions with the turbulence variation plotted as standard deviation. Besides a core with lower velocities, which was also visible in Figure 3-2, the distribution of the turbulent velocities can be seen in relation to the mean velocities. The direction correspond to the directions shown in Figure 2-4.

The measurements show an asymmetric profile. As this asymmetry is not expected to be present in a perfect set-up, it is assumed that it is caused by the propeller gearbox, which is mounted in the thruster duct or by inaccurate alignment of the propeller. Another possible inaccuracy in the measurements are the velocities in lateral direction as measured in the horizontal plane (centre top figure). It shows an average velocity in positive lateral direction, while it is expected to be approximately equal to zero similar to the vertical velocity in the vertical plane which does show the expected profile. The might be the result of circulation of the flow due to the limited width of the basin.

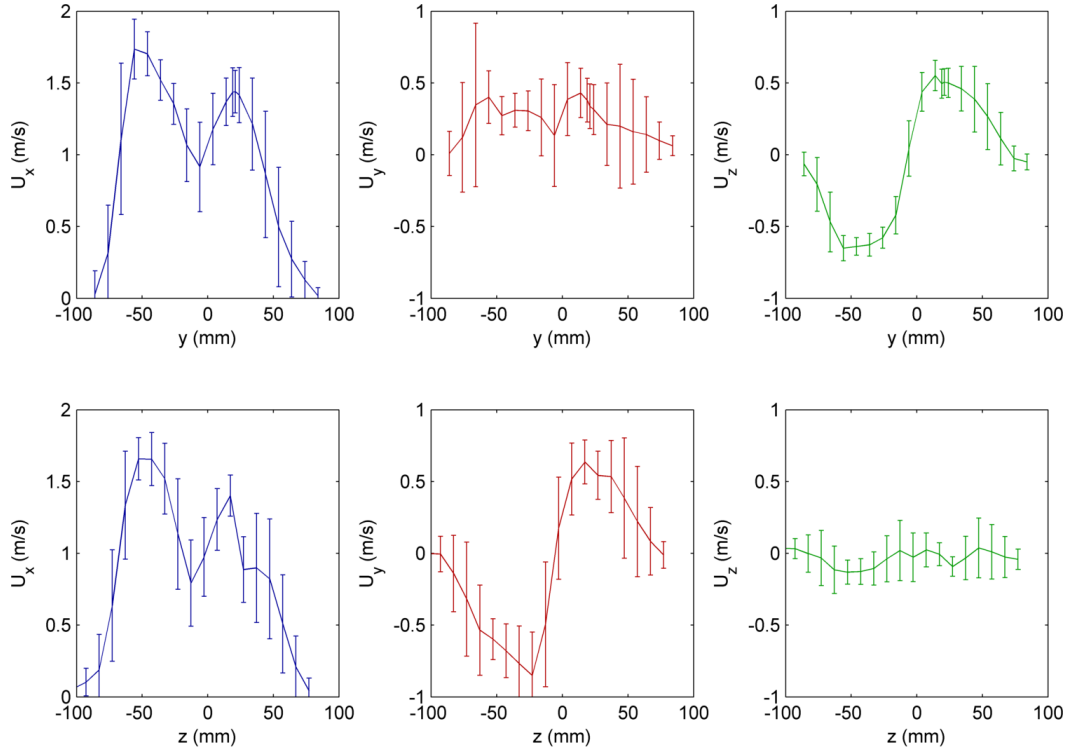


Figure 3-4: Velocity measurements of Van Doorn, showing the standard deviation all three directions for the horizontal plane (top) and the vertical plane (bottom)

3.1.2 CALCULATION OF THE THRUST AND TORQUE

The thrust and torque generated by the Vetus propeller of Van Doorn can be calculated based on the measurements. The closest measured points to the propeller were the measurements given in Figure 3-4, located at a distance of 95 mm of the outflow ($\frac{x}{D} \approx 0.9$) and approximately 245 mm from the location of the propeller. Due to friction and turbulence a part of the thrust and torque generated by the propeller will already be dissolved, but the values will be used as a first estimate. The data points at this axial location are shown at the left of Figure 3-5. As can be seen at the location of the measurement points at the left of this figure, the measurement locations in vertical direction by Van Doorn were not properly aligned with the bow thruster. This was also conceded in his report, but no new measurements were done at the efflux. Although this will influence the thrust and torque calculations, the comparison with the calculations in the horizontal plane will ensure this error to be small.

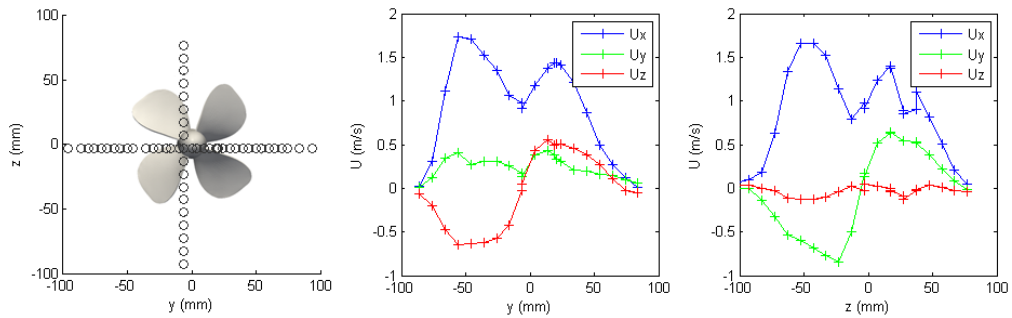


Figure 3-5: Location of Van Doorn's measurement points and the velocities at those locations

As the data shows obvious inaccuracies the calculation of thrust and torque has a large inaccuracy as well. Integration is done after removing of some errors in both x- and z-direction with Equation (2.15) and (2.16) resulting in the thrust and torque of Equation (3.4) and (3.5).

$$T = \rho A u_x^2 = 2\pi \cdot \rho \int u_x^2 \cdot r \, dr = 3 \cdot 10^1 \, N \quad (3.4)$$

$$Q = 2\pi \cdot \rho \int u_\theta^2 \cdot r^2 \, dr = 1 \cdot 10^{-1} \, Nm \quad (3.5)$$

Using these values, an initial guess of the axial and tangential parameters is made. Besides the thrust, the propeller radius and hub radius are entered in the equations of Chapter 2.2. The propeller radius is assumed to be equal to the radius of the duct, the hub radius is derived from Figure 3-3 and is approximately 18 mm. The coefficients can now be calculated with both Equation (2.10) and (2.11) and Equation (2.17) and (2.18). This results in the coefficients shown below.

Goldstein optimum coefficients:

$$\begin{aligned} A_x \cdot \Delta &= 1 \cdot 10^4 \\ A_\theta \cdot \Delta &= 5 \cdot 10^2 \end{aligned} \quad (3.6)$$

Ducted propeller coefficients:

$$\begin{aligned} A_x \cdot \Delta &= 2 \cdot 10^5 \\ A_\theta \cdot \Delta &= 7 \cdot 10^3 \end{aligned} \quad (3.7)$$

The thrust of Equation (3.4) equals a uniform efflux velocity of 1.59 m/s, which differs from the efflux velocity of 1.52 m/s which was concluded by Van Doorn. This difference might be the result of Van Doorn using a different (possibly incorrect) calculation of the integral or by Van Doorn not ignoring incorrect measurement data.

Based on his efflux velocity Van Doorn calculated the thrust coefficient with Equation (2.19) and concluded the coefficient to be 0.26 instead of the 0.28 as proposed by the manufacturer. The newly calculated efflux velocity proves that the original thrust coefficient was in fact correct.

The thrust and torque can also be calculated with the thrust and torque coefficient by using Equation (2.4) and (2.5). The thrust coefficient of 0.28 is used and the torque coefficients is estimated to be in the usual order of 0.05 [Triantafyllou & Franz, 2003]. This results in a thrust of 11 N and a torque of 0.23 Nm. This does not seem to have any relation to the thrust generated in Equation (3.4) and (3.5) and will therefore not be further applied.

3.1.3 CHANGE IN TIME

It is possible that the individual blades of the propeller result in extra velocities as was also visible in the cavitation of the propeller blades in Figure 2-5. To check if these velocities show in the measurement results of Van Doorn, a closer look was taken into the data by using the raw data instead of the time-averaged data used in other measurements.

The six-bladed propeller had a rotation rate of 1021 rotations per minute which equals 102 passing blades per second. The used ADV measurement device was only able to record with a frequency of 25 Hz and therefore makes it impossible to measure those fine oscillations in the efflux. This makes the measurements approximately uniform. This justifies the use of a steady force at the location of the propeller.

The measurements of an ADV device cannot simply be converted to an average and a standard deviation as might be expected. Instead a filter has to be applied to remove the error induced by measuring the wrong reflection of the seeded particles. Depending to the amount of filtering the mean velocity and, to a greater extend, the turbulent variation change. In Figure 3-6 the variance in time is shown for both the full recording and a close-up of the start. As an arbitrary but close point the location of x +94 mm, y +14 mm and z -2.8 mm direction relative to the axis at the outflow of the bow thruster is taken.

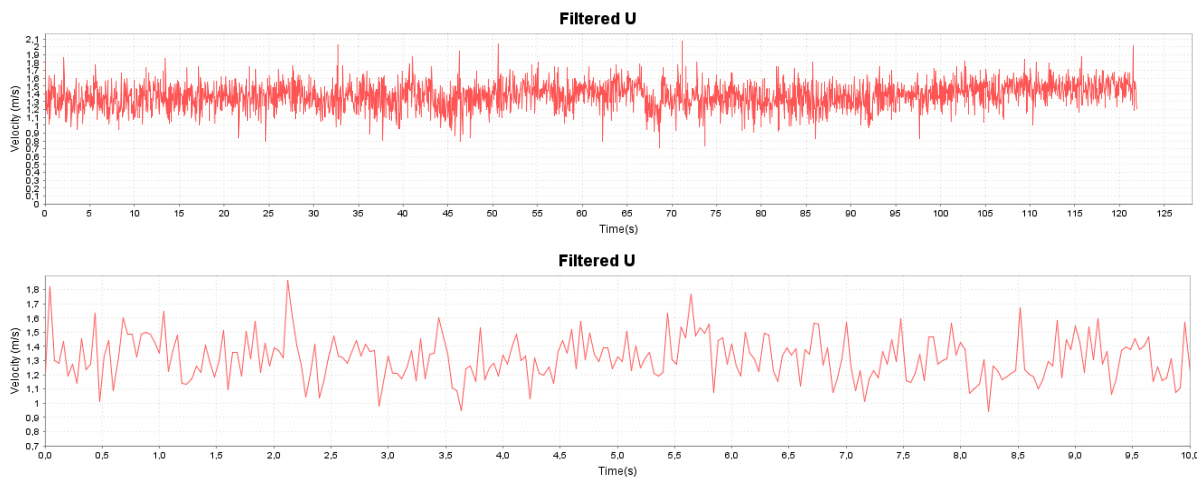


Figure 3-6: Measured velocities after filtering with an exclude factor of 4.0 for the full two minutes (top) and a close-up of the first ten seconds (bottom)

Table 3-1 shows the velocities and turbulence in three direction for a couple of ‘Exclude’ factors. Measurements which deviate more than the specified factor of the standard deviation are excluded. It can be concluded that the mean velocities hardly change with a higher exclude factor, but the standard deviation (or turbulent velocities) changes significantly but also seem to converge for higher exclude factors. Comparing the values to the case of Van Doorn shows his results are comparable an exclude factor of 4.0, for which only the larger inaccuracies have been removed. This should be kept in mind when comparing to the model, as the chosen filtering could also result in other values of the turbulent energy.

Table 3-1: Sensitivity analysis of velocities and turbulent velocities to the *Exclude* filtering coefficient in [m/s] for an arbitrary measurement of 2 minutes.

Values	Exclude = 1.0	Exclude = 2.0	Exclude = 3.0	Exclude = 4.0	Van Doorn
$U_{x,m}$	1.38	1.38	1.38	1.37	1,37
u_x	0.09	0.14	0.16	0.16	0,17
$U_{y,m}$	0.43	0.43	0.43	0.43	0,43
u_y	0.09	0.15	0.17	0.18	0,17
$U_{z,m}$	0.55	0.56	0.56	0.56	0,55
u_z	0.05	0.09	0.10	0.10	0,11

3.2 MEASUREMENTS AT QUAY WALLS

The behaviour of a water jet on vertical quay wall differs from the deflection at an open quay wall, as was already shown in chapter 2.3.4. Besides an upward and lateral spreading of the water jet, also a part of the water jet is redirected downward.

Measurements at a quay wall were done in a physical model at the TU Delft. A propeller slightly smaller than the propeller used in the research of Van Doorn was used ($D_p = 100$ mm) and exerted on a wall at a distance (quay clearance) of 500 mm. A limited number of measurements were initially done with an EMS under the vessel and between the vessel and the quay [Van der Laan, 2005] [Nielsen, 2005]. In later research more measurements of the outflow of the bow thruster were needed for the building of numerical model and the more detailed ADV was used to gather data over the full height between the quay wall and the ship and for measurements at the outflow at the propeller axis [Van Blaaderen, 2006]. A full overview of the setup and the measurement locations of Van Blaaderen are given in Appendix C.2. It can be seen that the model ship was fixed to a side wall of the basin, which will likely have resulted in circulation or obstruction of the flow.

In contrast to other data, the measurements by Van Blaaderen did not show a collapse of the core in the outflow of the bow thruster (Figure 3-7). Unlike any other case the core remains evident and the water flow is further separated at the quay wall. It was not concluded what the reason was of this remarkable event, but possible parameters of influence are the limited distance to the quay or the low flow velocities. It is also possible that inaccuracies were generated with the use of the ADV. Reproduction in a physical or numerical model would be needed to confirm or reject this remarkable occurrence.

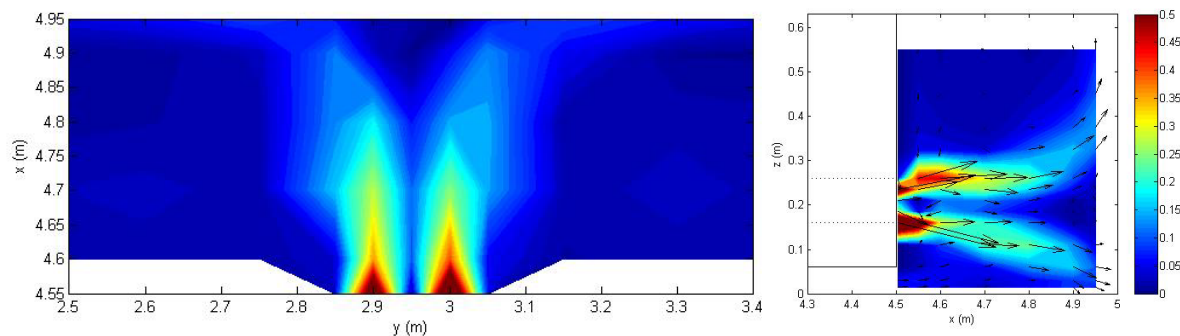


Figure 3-7: Measurements of the outflow in vertical direction at a closed quay wall in [m/s] in the horizontal plane (left) and vertical plane (right) [Van Blaaderen, 2006]

3.3 EVALUATION OF FORMER NUMERICAL MODELS

Besides physical modelling several numerical models have been developed at the TU for the simulation of bow thrusters. The intended application of these models differed from a full calculation of the induced velocities to an estimation of the necessary basin for the physical model. All three models were created in PHOENICS and were successors of each other.

At first a model was generated based on the case set-up and measurements of Figure 3-1 [De Jong, 2003]. In this model the water jet was, despite the conclusions of Figure 3-2, modelled as a pressure jet and a *spin* was added for tangential velocities. The core was implemented by adding a circular core with a width of $0.3D$. Calibrated on the measurements for this set-up, it was subjected to a quay wall (Figure 3-8).

Although this model gave similar results in the outflow of the water jet, it was doubtful if results were reliable. This caused by to the lack of measurement data and the reliability of these data. Other reported flaws in the numerical model were the lacking of propeller blades, which would induce extra turbulence, and the inaccuracy of the circulation when using the $k-\epsilon$ turbulence model.

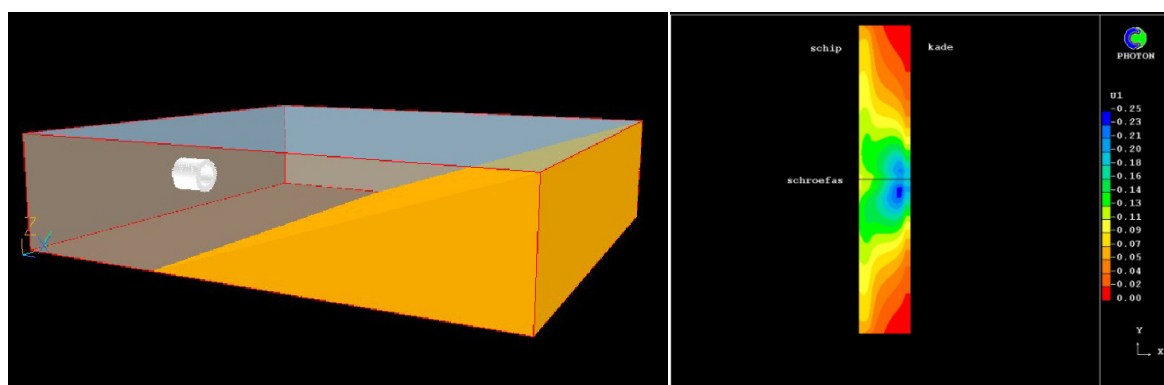


Figure 3-8: (left) Modelling of the flow on a slope; (right) Resulting flow velocities for flow at quay wall [De Jong, 2003]

After measurements at a closed quay wall the model was calibrated, but still showed unsatisfactory results [Van der Laan, 2005] [Nielsen, 2005]. A comparison between measurement results is shown in Figure 3-9. In the top of this figure the measurement data is shown with the visual measurements in blue and the measurements with the EMS in orange. The bottom figure shows the velocity vectors of the numerical model.

It can be seen that the velocities close to the location of impact of the water jet at the quay wall show a good correlation between the physical and numerical model. Further from this point the lateral velocities are higher than measured and at the boundaries of the basin a large error can be noticed. In addition to those errors, also the circulation under the ship as observed in the physical scale model is not correctly modelled in the numerical model.

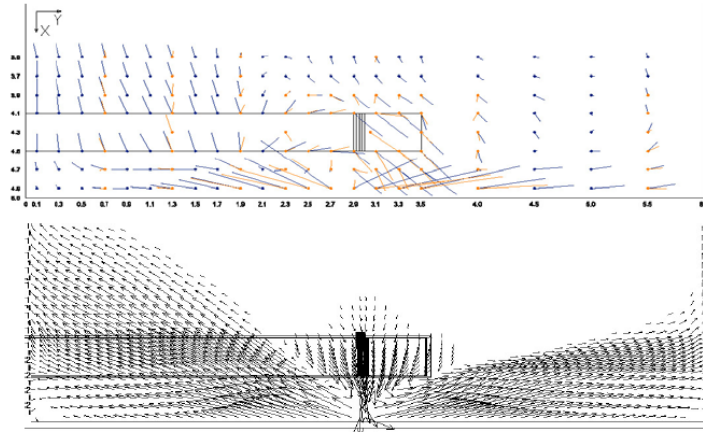


Figure 3-9: Comparison of the physical (top) and numerical (bottom) scale model at the bottom of the basin [Nielsen, 2005]. In the physical scale model both the visually observed (blue) and the measured (orange) velocities are shown

Continuing these studies, Van Blaaderen started with an analysis of the behaviour of the numerical model. This also included a sensitivity analysis in which it was concluded that it is important to model the outflow of the bow thruster correctly for a reliable numerical model.

The sensitivity analysis also included the variation caused by the use of different turbulence models and wall functions. Besides the regular $k-\varepsilon$ turbulence model, adaptations by Lam-Bremhort and Chen-Kim were verified. For the wall functions general logarithmic wall function, an equivalent roughness parameter and a fully rough wall-function were applied. From the generated flow pattern it was concluded that there was only a minor dependence on these models and wall-functions and that the initial chosen $k-\varepsilon$ model and wall functions were justified for further use.

To be able to improve the outflow, Van Blaaderen did additional measurements with an ADV at mainly the quay clearance, the distance between quay and vessel. As concluded from Figure 3-7 a clear core in the outflow of the bow thruster can be noticed that does not collapse after a distance of 2 to 3 times the diameter as written in literature and shown in earlier studies. This was included in the PHOENICS model by adding a core plate of 0.85 times the diameter of the propeller duct and including an extra turbulence source at the screw (Figure 3-10).

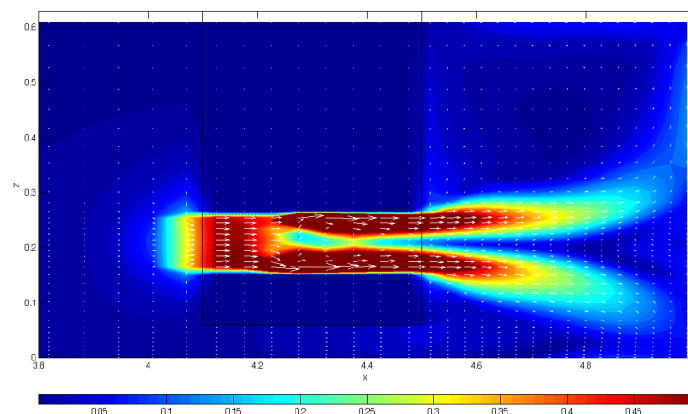


Figure 3-10: Calculated vertical velocity field with a core plate of 0.85D [Van Blaaderen, 2006]

However, as discussed in chapter 3.2, the existence of this large low-velocity core is doubtful and is not found in earlier measurements. The model also only proved to be correct in the direct outflow for which it was calibrated, but results at other locations were less accurate.

3.4 CONCLUSION - NECESSITIES NUMERICAL MODEL

From *Chapter 3.1* it shows that the research done by Van Doorn [2012] shows a large and complete set of measurements for different scale models of open quay structures. Between the different scenarios the slope angle, water depth, roughness of the slope, distance between the thruster and the slope and the presence and alignment of these piles were varied. The basin width might have resulted in circulation and limited the scenarios to the steeper slopes. Both at the slope as at the efflux of the bow thruster measurements were done, making it an ideal case for the calibration of a numerical model. The measurements are done in three directions with an ADV and did not show any change over time for the used measurement frequency.

Chapter 3.2 shows that at a vertical quay wall a limited series of measurements was done by Van Blaaderen. These measurements also include most of the necessary locations for the calibration of a model, but are only done for one scenario. As the model ship was fixed to a side wall of the basin, this is likely to have resulted in circulation or obstruction of the flow. Besides this expected error, the measurements also show an unexpected large low velocity core at the intersection of the propeller axis with the quay wall.

Besides physical scale models, also numerical models were constructed in previous researches and an overview of those is given in *Chapter 3.3*. The bow thruster is implemented in different ways to model the water jet in the close proximity of the efflux as good as possible, but all show a large deviation to the physics of a propeller jet and a deviation to the measurements. The best results were obtained by adding a wide core plate of $0.85 D_p$ and an arbitrary additional spin to a uniform introduced water jet, but these additions were arbitrary and although the numerical model agreed well with the measurements at the efflux, it deviated from the measurements in order regions.

Instead a numerical model needs to be created that more closely approximates the outflow and thereby radial diffusion of the water jet. It is expected that a correct modelling of the outflow will result in a good simulation of the flow velocities in the water jet.

4 APPLICATION OF A NUMERICAL MODEL

For the numerical model a software package fulfilling all demands is found in OPENFOAM. This chapter covers the introduction of OPENFOAM (*Chapter 4.1*), the implementation of the actuator disc (*Chapter 4.2*) and the set-up of the numerical scale model within OPENFOAM (*Chapter 4.3*).

In *Chapter 4.4* the implementation of the body force method is tested and a comparison is made between the *Goldstein* and *ducted* distribution function. The body force coefficients as derived for the measurements of Van Doorn are used to compute efflux velocities, which are compared and calibrated to the measurements. The calibrated model is compared to both the theoretical water jet of Blaauw and Van der Kaa and the measured velocities of Van Doorn for several scenarios.

4.1 OPENFOAM EXPLAINED

As chosen modelling package to implement a bow thruster and compute the resulting outflow, the open source code OPENFOAM (version 2.2.2) is used. Appendix D shows other packages that were considered for this thesis.

OPENFOAM stands for Open Field Operation And Manipulation and is a free and open source CFD software package produced by OpenCFD Ltd since 2004. It is protected by the OPENFOAM Foundation and is being used in most areas of science and engineering. OPENFOAM is written in the C++ programming language and includes a wide variety of solver applications. Pre- and post-processing is possible with both integrated applications as well as effective communication with third party (open source) software.

The computation of the water jet in OPENFOAM is solved by computational fluid dynamics (CFD). CFD is the analysis of systems involving fluid flows by means of computer-based simulation. The Navier-Stokes equations describing the link between pressure and velocity are discretized to a limited number of equation by dividing the total volume in a finite number of control volumes. In OPENFOAM's discretization scheme, called the finite volume method, the centre point of these volumes is used together with the flux over the faces between the cells to iteratively compute the transient or steady state solution.

As the storing of all variables in the centre point is by itself an unstable finite volume approach, the usual strategy is to store the volume based quantities in the centre point of the cells and the flux based quantities on the faces. This is called a staggered grid [Versteeg &

Malalasekera, 2007]. In OPENFOAM a different strategy is applied, which is called a co-located grid, where all quantities are stored at the same nodes (the centre points). For a co-located a special treatment is used for the pressure to avoid the pressure/velocity decoupling, in the spirit of the Rhie-Chow correction [Rhie & Chow, 1983] [Kärrholm, 2006]. This method was not invented to take care of sudden pressure jumps or discrete body forces [Réthoré & Sørensen, 2008]. To overcome an arising problem, the body forces can be smoothed over a wider area [Mikkelsen, 2003] or a modification of the Rhie-Chow algorithm can be applied [Réthoré & Sørensen, 2008].

Besides the absence of software costs, one of the main advantages of OPENFOAM is the freedom it offers for all sorts of applications. Depending on the wishes of the user, a solver is chosen with the desired algorithm of the Navier-Stokes Equations (e.g. SIMPLE, PISO or PIMPLE), the desired time-dependency (steady state or transient) or the use of a free water surface (with the Volume of Fluid method). For the algorithms of the solver the desired integration method for each term of the equations needs to be specified together with the maximum and relative tolerances and relaxation factors, which help preventing instabilities. As the code is fully open and editable, any further wishes can be implemented directly in the OPENFOAM code.

The official documentation for OPENFOAM is very limited and it has no graphical user interface, making the necessary parameters definitions sometimes hard to figure out. For the using of the more advanced features, investigation the code and reading discussions and reports of the community are the best sources.

4.2 IMPLEMENTATION OF THE BOW THRUSTER

The propeller of the bow thruster can be included in OPENFOAM in different ways. The first option is to model the geometry of the propeller in detail and use one of the options provided in OPENFOAM to rotate the propeller. The second option is to replace the propeller with an approximation of the forces generated with the propeller by use of the actuator disc method. Both methods are elaborated below.

4.2.1 MODELLING OF THE PROPELLER

The first and most obvious solution is the inclusion of an exact model of the rotating propeller in the simulation. This requires the shape of the propeller to be known in great detail and also needs possibilities for the mesh to rotate. OPENFOAM provides several options for a moving mesh to be incorporated in the simulation [Petit, 2007].

- Single Rotating Frame (SRF). This feature makes it possible to use a stationary mesh, but add a fictitious rotation to the propeller. The propeller will not actually move in the mesh, but the rotational component is included in the solver. It can be used within most steady state solvers.
- Multiple Reference Frames (MRF). This feature is comparable with SRF with the addition of being able to connect the fictitious moving mesh to a stationary mesh.

- Arbitrary Mesh Interface (AMI). This is a method in which the propeller is actually moving through the grid. It uses dynamic meshing in which it is possible to rotate a part of the mesh and projecting the boundaries on the stationary grid.

Of these options both MRF and AMI provide the options needed for creating the case that will be reconstructed, MRF being the choice for steady state solutions and AMI for transient solvers. However, for both options the actual shape of the propeller has to be accurately known. Three-dimensional drawings of those are often protected with care by the designers and the creation of an exact replica will take a lot of effort. In case of an incorrect velocity field the propeller cannot be simply adjusted. Another downside of these methods is the high computational running time that is required for this necessary detailed meshing. A level of detail at a location that is not necessary for reaching the objective of this thesis.

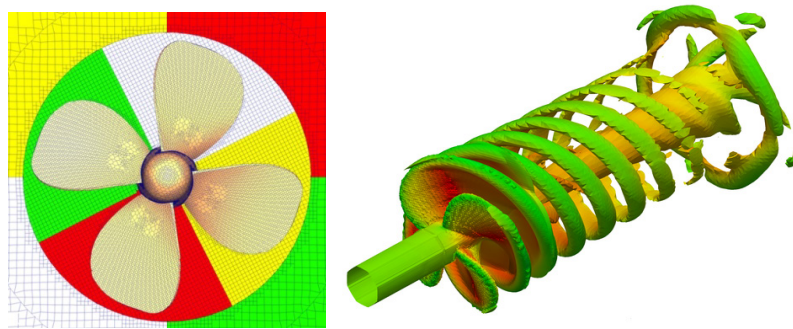


Figure 4-1: Use of an Arbitrary Mesh Interface with a propeller in OPENFOAM ⁴

4.2.2 IMPLEMENTATION OF AN ACTUATOR DISC

A better option is to not include the propeller in the mesh, but to use an approximation of the forces generated by the propeller. A less detailed mesh is needed which gives a significant reduction of the computational costs.

The method that was already earlier introduced in Chapter 2.2 and which is often applied as a replacement of ship propellers and wind turbines, is the replacement with an actuator disc. At the location of the propeller this generates a steady body force in axial and tangential direction as a function of the radius to the propeller axis. The approximation based on the Goldstein optimum, which is also included the marine CFD software FineTM/Marine and CFD SHIP-IOWA, is already presented in Chapter 2.2.1 and a similar approach is done for the implementation for a ducted propeller in Chapter 2.2.2 [FINE(tm)/Marine, sd] [Paterson, et al., 2003].

Implementation in the C++ code has been made for different solvers and different OPENFOAM versions. As the basis of the implementation is similar for all solvers, the inclusion in the, later discussed, simpleFoam solver is given in Appendix E for the Goldstein optimum.

At first the location of the propeller is determined based on the defined parameters in the propeller dictionary, as it allows for the implementation of one propeller in any direction at

⁴ <http://www.openfoam.org/version2.1.0/ami.php>

any location within the mesh. To achieve that, the grid is temporary rotated to the default situation where the propeller is directed in x-direction. This is done by rotating each point about the axis $\langle v_1, v_2, v_3 \rangle$ by the angle φ . When the rotation axis is a unit vector this leads to the (implemented) rotational matrix shown in Equation (4.1) [Murray, 2013].

$$\begin{pmatrix} v_1^2 + (1 - v_1^2) \cos \varphi & v_1 v_2 (1 - \cos \varphi) - v_3 \sin \varphi & v_1 v_3 (1 - \cos \varphi) + v_2 \sin \varphi \\ v_1 v_2 (1 - \cos \varphi) + v_3 \sin \varphi & v_2^2 + (1 - v_2^2) \cos \varphi & v_2 v_3 (1 - \cos \varphi) - v_1 \sin \varphi \\ v_1 v_3 (1 - \cos \varphi) - v_2 \sin \varphi & v_2 v_3 (1 - \cos \varphi) + v_1 \sin \varphi & v_3^2 + (1 - v_3^2) \cos \varphi \end{pmatrix} \quad (4.1)$$

To prevent pressure/velocity decoupling, two options were described in Chapter 4.1. The changing of the pressure correction would approximate a possible analytical solution most closely [Réthoré & Sørensen, 2008]. This would require a lot of mathematical and programming effort and as a slight change in resulting thrust and torque can be corrected by calibrating the coefficients it was chosen to reduce the pressure/velocity decoupling problem by implementing the actuator disc over a thickness of multiple cells which can be specified by the user. This thickness also ensures that cell centres are covered over the full radius. As the resulting numerical thickness will differ from the exact defined thickness, the difference will be outputted to the user when running the solver, to make the user aware of the difference in implemented body force.

The radius to the axis of the propeller is computed and the body forces for each cell are determined. These are added to the Navier-Stokes equation. Simplified for a Newtonian fluid this is given in Equation (4.2). In this equation the vector F is the total body force consisting of both the axial and tangential forces in a Cartesian vector.

$$\underbrace{\frac{\partial U}{\partial t}}_{\text{change in time}} + \underbrace{(U \cdot \nabla)U}_{\text{convection term}} = \underbrace{\nabla(v \nabla U)}_{\text{diffusion term}} - \underbrace{\frac{1}{\rho} \nabla p}_{\text{source term}} + \underbrace{\frac{F}{\rho}}_{\text{body force term}} \quad (4.2)$$

The use of the new solver *simpleFoamProp* requires several new parameters to be defined in the propeller dictionary. These parameters need to be specified for each simulation, allowing flexibility and calibration. An explanation of the dictionary parameters is given in Table 4-1.

Table 4-1: Parameters used in the propeller dictionary

Parameter	Description	Example
propOrigin	The location on which the origin of the propeller is centred	(0.060 0 0)
outflowDirection	A vector indicating the direction of the propeller axis	(1 0 0)
coreRadius	The radius of the hub [m]	0.018
radius	The radius of the propeller [m]	0.055
Ax	The coefficient multiplier for the axial direction	1.87e6
Atheta	The coefficient multiplier for the tangential direction. The positive direction is defined with the right hand rule around the outflowDirection	-6.00e4
rho	The density of the water [kg/m ³]	1e3
thickness	The (numerical) thickness of the actuator disc [m]	0.0075

To prevent circulation of the flow within the thruster duct, a simple hub is included in the model. It turned out that a smooth shaped hub closely approaching the shape found in propel-

ler design gave the best results. This shape is shown in the meshing stage in Figure 4-4. It is not located in the exact centre of the duct. Early drawings of the model set-up by Van Doorn suggested that he used a location more to the front of the duct, so it was located 60 mm from the centre in x-direction.

At the hub and tip of the propeller also a large vorticity could be noticed as shown in the cavitation in Figure 2-5. This could be implemented in a similar method as the local momentum increase, by locally increasing the turbulence kinetic energy at those locations. This modification is not done in the solver, but should be done by modifying the turbulence model. However, the implementation of the turbulence model in the OPENFOAM code prevents a simple relation of the location within the grid to the kinetic energy, making a larger change to the model necessary. For applying this change insufficient knowledge and time was available.

The momentum increase is applied in a fixed duct which prevents the increase of the discharge at that location. This in contrast to the usual application for wind turbines and free propellers. This prevents the momentum to show a sudden increase at the location of the actuator disc, when no sudden velocity jump is corrected in the continuity equation.

This correction is not included, as it is expected that the body force results in the velocities to increase already at the inflow of the duct. When correctly simulated, both the inflow and the outflow of the duct would have the same discharge, which is necessary for comparing for cases where circulation is expected.

Another method, to prevent the implementation within the grid, would be to add the actuator disc at the side of the mesh. In this method the velocities are directly defined in the boundary conditions with the help of special OPENFOAM utilities. An explanation of how this method would work is given in Appendix G. The downside of this implementation is the need for large adjustment of the grid at the location of the thruster.

4.3 MODEL SETUP

As concluded in Chapter 3.1, the scenarios of Van Doorn were detailed and suitable for reconstruction in a numerical model. The dimensions of the scenarios, as shown in Appendix C.1.2 are used for the recreation of this scale model in a numerical scale model. Besides the discretisation of this domain in the meshing stage, other numerical aspects need to be specified.

A choice needs to be made for the turbulence model, where a balance needs to be made between the accuracy of the results and the computational costs. The boundary conditions and initial conditions need to be determined and the algorithms used for the solving of the computational domain are given.

4.3.1 TURBULENCE MODELLING

In CFD modelling the Navier-Stokes equations are solved for the domain in an iterative process. For turbulent flows this requires a very fine numerical grid when full computation of the turbulent fluctuations is done, which would result in an unrealistic high computational costs

for this case (DNS). Instead the turbulence is approximated with a turbulence model. In Appendix A full elaboration of the possibilities is given in which it becomes clear that a higher accuracy also results in higher computational costs. The category of turbulence models that is used, is called Reynolds averaged Navier-Stokes (RANS) or Reynolds averaged simulation (RAS). In this category the Reynolds stresses are added to the equations as a source term and the Boussinesq approximation is applied to included the small turbulence as an increase in the viscosity. The turbulent fluctuations are no longer modelled, but an average is computed.

A large variety of RANS models exists and is shown in Appendix A.1. One of the often used models is the $k-\varepsilon$ model which is most widely used and validated. However, this model has shown to be inaccurate for round jets. An adjustment to this model has been made to the realisable $k-\varepsilon$ model, improving its performance for swirling flows, flow separation and round jets, with a lower stability as disadvantage. This realisable $k-\varepsilon$ model will be used in the simulation [Shih, et al., 1994].

Near walls the modelling of a high-Reynolds-number model like the $k-\varepsilon$ models cannot be solved and the wall needs to be separately included in the turbulence model. This region can no longer be approximated with the realisable $k-\varepsilon$ model as only low-Reynolds-number models are able to predict the laminar flow in this region. The turbulence equations are no longer solved, but approximated with the assumptions of constant shear and equilibrium between production and dissipation of the turbulence quantities.

4.3.2 MESH CREATION

For an accurate numerical solution, the quality of the grid is of great importance. OPENFOAM offers a lot of flexibility by allowing an unstructured polyhedral grid, but although this offers unlimited possibilities, the quality of the mesh needs to be ensured. Three meshing methods are compared within OPENFOAM and for all three a short introduction is given and the result for the cross-section of the duct is shown in Figure 4-2.

The most basic implemented mesh generation is done with the BlockMesh utility. In BlockMesh different blocks can be combined for the generation of simple meshes. In these blocks a grid of hexahedral cells is generated using a uniform or gradient grading. Although a high level of control is possible, it offers limited freedom as higher grading in a single area automatically results in the expansion of grid cells in a larger area. The skewed cells at the transition of the corner of the square to the circle also resulted in numerical instability.

The second method available is the possibility to import a full mesh from another package. Most of the frequently used meshing utilities can be converted to the OPENFOAM format. A good example of an unstructured grid, generated with Salome, is shown in the centre of Figure 4-2, but this grid results in numerical diffusion.

As a last method the integrated SnappyHexMesh mesher is introduced. It offers the combination of using predefined elements and introducing them in the BlockMesh grid. It allows for the refinement of the grid in specific areas and the introducing of extra wall layers. This im-

plies that also the hexahedral grid is used, which offers the best solutions in longitudinal direction for the convection. The flexibility and quality of the mesh generated by SnappyHexMesh made this the chosen method for the construction of the open quay wall scenarios.

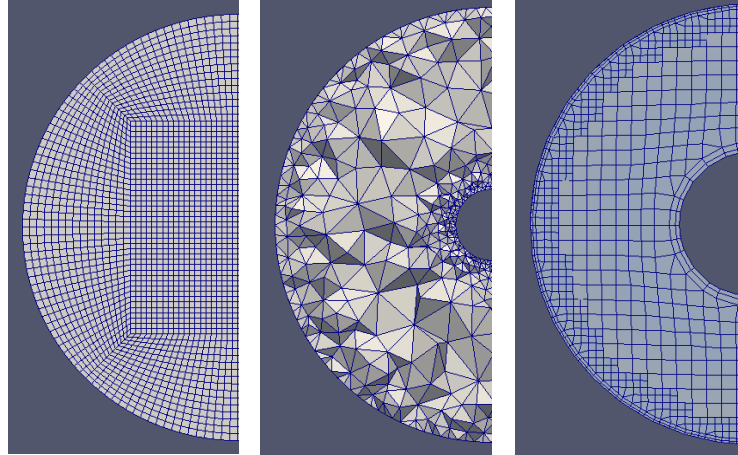


Figure 4-2: Examples of mesh generated by blockMesh (left), Salome (center) and snappyHexMesh (right)

All three-dimensional objects were created in an external program (Salome Platform) as individual objects. This offers the greatest freedom in the specification of the near-wall layers for each object for a correct use in the near-wall model. This specification is needed to make sure the first layer of the cells is still in the region which is in the computable fully turbulent sub-layer.

The non-dimensional wall distance for a wall-bounded flow (y^+) is defined in Equation (4.3), where u_* is the friction velocity at the wall, y is the distance to the nearest cell and ν is the local kinematic viscosity of the fluid.

$$y^+ = \frac{u_* y}{\nu} \quad (4.3)$$

For a correct approximation of the near-wall area, the mesh needs to have y^+ values within a certain range for all important regions. The y^+ value should not be too low, as a point close to the wall would still be in the viscous sub-layer which cannot be accurately modelled in a model with high Reynolds numbers, nor should it be too high as the presence of a wall function will not be taken into account. A y^+ of 30 – 100 is often assumed to offer proper results.

SnappyHexMesh offers two features to locally refine the grid and both are used to have the y^+ reduced to the specified range. It is possible to refine a certain region to cells which are a fraction of the base cells size. This is applied in several steps of refinement in the region close to the outflow to have both a fine mesh at the high velocity locations and a coarse mesh at a further distance from the efflux of the bow thruster to reduce the computational costs. The second feature is the addition of wall layers, which is applied near all structures to reduce the y^+ values [OpenFOAM Foundation, 2013].

To reduce the computational costs some further simplifications are done. The numerical domain is reduced to a width of 3 metre to still include the lateral flow over the slope and the

number of piles is reduced to just a number three or four (depending on the configuration) rows of piles close to the outflow.

In Figure 4-3 and Figure 4-4 the grid for Scenario 6 is shown. Marked in the figures are the areas where the refinement is increased. The dimensions of all elements of the grid are shown for the physical scale model in Appendix C.1.2 and the cell size for the different refinement areas is shown in Table 4-2. The cell size is reduced for areas closer to the bow thruster and has the finest grid near the structures, where also the additional wall layers are visible.

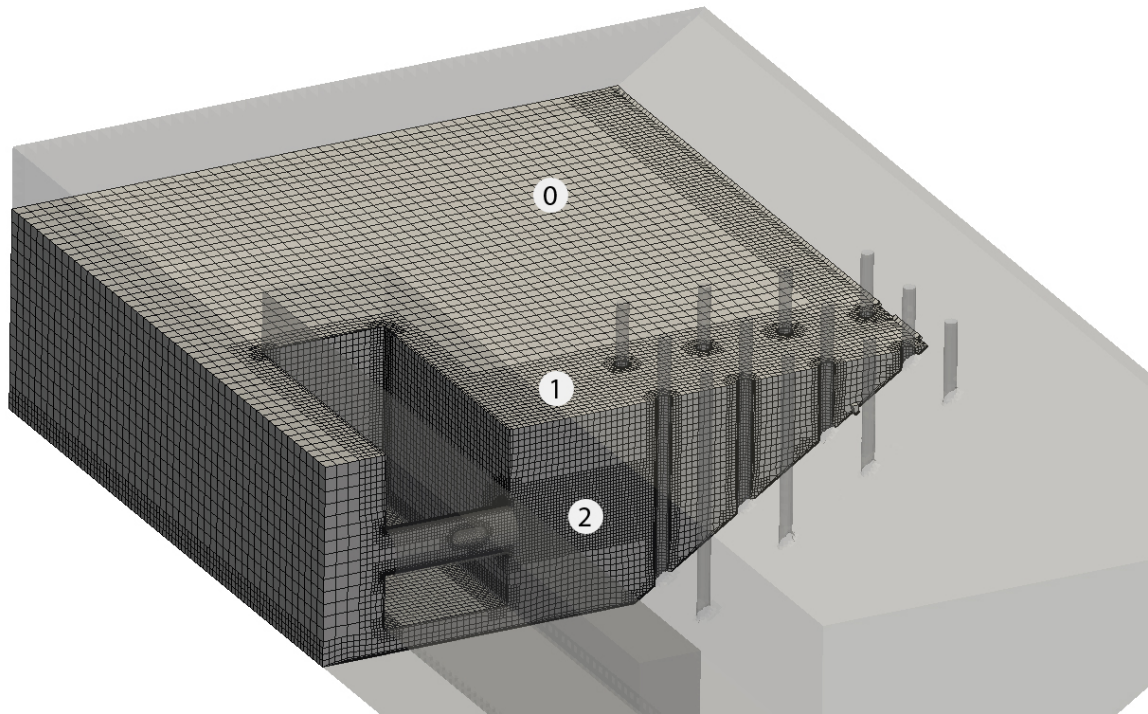


Figure 4-3: Half of the grid for scenario 6 of Van Doorn. The numbers show the level of refinement

At the right of Figure 4-4 the grid at a pile is shown. It can be noticed that the added extra layers disappear at the water surface boundary of the piles, but at those locations the low flow velocity already results in sufficient low y^+ values.

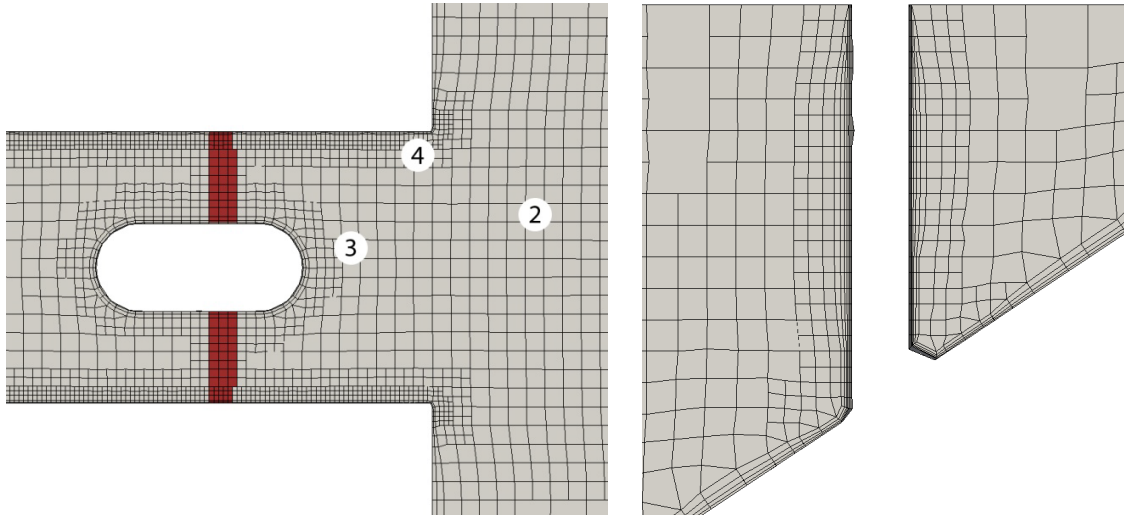


Figure 4-4: Close-ups of the grid at the propeller with the actuator disc shown in red (left) and the at a pile (right). The numbers show the level of refinement

The grid shown in these figures is to the upmost extend constructed of hexahedral cells, combined with prism and polyhedral layers near the structures. In total it consists of $6.5 \cdot 10^5$ cells and a combined total of $2.23 \cdot 10^6$ internal and external faces.

Table 4-2: Size of cells for the different refinement levels

Refinement level	Cell size
0	30 mm
1	15 mm
2	7.5 mm
3	3.8 mm
4	1.9 mm

4.3.3 BOUNDARY CONDITIONS AND INITIAL CONDITIONS

Boundary and initial conditions need to be specified for, respectively, the boundaries and the internal mesh. Both the velocity (U) and pressure (p) need to be specified, although these boundaries are usually linked through the Navier-Stokes equations in other CFD software, where specification of both is not necessary. Boundary conditions will be discussed to couple these boundaries also in OPENFOAM. For the discussed $k-\epsilon$ turbulence model both variables need to be determined as well and at last the turbulent kinematic viscosity (ν_t) needs to be specified even though this is already coupled to the turbulence variables. This is necessary, as the alteration that is usually added to the momentum equations in the near-wall region for inclusion of the resistance, is added through the turbulent kinematic viscosity instead in OPENFOAM.

As will be explained in the next paragraph, a steady state case is used. As only the converged model is used in steady state solutions, the initial conditions are not of importance. For every initial situation the model should converge to the same solution, but the initial conditions can be adapted to increase the speed of the numerical simulation.

Unlike the initial conditions, the boundary conditions are of great influence to the solution and need to be chosen with care. Table 4-3 shows the boundary conditions as used in this simulation showing the two basic boundary conditions known as the Dirichlet condition ($\Phi = \text{Constant}$) and the Neumann condition ($\partial\Phi/\partial x_j = \text{Constant}$). OPENFOAM offers options to combine those conditions, which is indicated as a criteria in the table.

A different class of boundary conditions are the computational boundary conditions. One possible computational boundary condition that could be used is a symmetry boundary condition, but although most of the geometries are symmetric, the rotating water jet requires a computation of the full domain.

A special remark is necessary to a few of the types included in table. At the outer domain, an unlimited length of the basin is simulated. The velocity inlet is specified with a Dirichlet condition of 0 m/s, but when the outward directed flux through the domain is positive, a Neumann boundary condition applies. The pressure at the outer domain boundary is coupled to the velocity by implying a constant energy head with Equation (4.4), where p_t is the generated boundary pressure and p_0 the reference pressure, which is zero. These boundary conditions proved to be a correct implementation of a outer domain boundary by comparing it to a case with a larger computational domain.

$$p_t = p_0 + \frac{1}{2}\rho|U|^2 \quad (4.4)$$

For the k, epsilon and nut solutions wall functions are used near the surfaces of the objects, with a no-slip condition for the velocities. The common wall functions are applied in the basic model, but as these do not allow for any change in roughness, they will be further tested in the sensitivity analysis.

Table 4-3: Boundary conditions

Property	Patch	Criteria	Type	Value
U	Outer domain	$U > 0$	Neumann	(0 0 0)
		$U < 0$	Dirichlet	(0 0 0)
	Water surface	U_x, U_y	Neumann	(0 0 0)
		U_z	Dirichlet	(0 0 0)
	Vessel, bottom, piles		Dirichlet	(0 0 0)
p	Outer domain		Dependent on velocity	
	Water surface		Neumann	(0 0 0)
	Vessel, bottom, piles		Neumann	(0 0 0)
k	Outer domain		Dirichlet	0.05
	Water surface		Neumann	0
	Vessel, bottom, piles		Wall function	
ϵ	Outer domain		Dirichlet	0.03
	Water surface		Neumann	0
	Vessel, bottom, piles		Wall function	
ν_t	Outer domain		Neumann	0
	Water surface		Neumann	0
	Vessel, bottom, piles		Wall function	

4.3.4 NUMERICAL PROCESS

As the default solver for most simulations, the adaptation to the simpleFoam solver is used for the solving of the Navier-Stokes equations. This solver is based on the SIMPLE algorithm for the steady state solution of a single-phase model [Patankar & Spalding, 1972]. In inverse order these three properties are explained.

The last mentioned property is the single-phase model. This means that the entire simulation consists of one specific fluid. This is contrary to the multiphase models, where the interaction of multiple substances can be included. In multiphase models an open water surface would be possible, but by choosing a single-phase model this is excluded as it is not expected to have a significant influence and as it does increase the computational costs. The visual observations of Van Doorn proved that only a small surface wave appeared at the start of the bow thruster but no other oscillations of the water surface were observed.

The second mentioned property is the steady state solution. This is one of the basics of the SIMPLE algorithm and means that the output of the model exists of one converged solution that no longer changes in time. Since the measurements of Van Doorn are all done in a steady flow and averaged over several minutes, this results in no reduction of the quality of the results to the calibration, but offers a great reduction in computational costs as the intermediate time steps are not accurately computed.

The SIMPLE algorithm (Semi-Implicit Method for Pressure-Linked Equations) is based on not fully resolving the pressure-velocity equations for an individual iteration step [Patankar & Spalding, 1972]. It consists of four steps in an iterative loop where successively the momentum equations are solved, the corrected pressure equations are solved, the velocities and pressures are corrected and the other transport equations are computed [Versteeg & Malalasekera, 2007]. Under-relaxation is used to reduce the changes per iteration and prevent the pressure correction equation from diverging. A correct choice of these case-dependent factors is essential as a large factor will lead to oscillatory or diverging solutions and a small factor will cause slow convergence.

For all equations solved with the SIMPLE algorithm the discretisation method of each individual term needs to be specified. The discretisation method can be valued by determining the conservativeness, the boundedness, the transportiveness and the accuracy. A scheme is consistent if the flux over the face is represented by one and the same expression in adjacent control volumes. Boundedness means that the nodal values are always within expected bounds. For a case without source terms this means that the property will be bounded by its boundary values. The transportiveness of a fluid flow is determined by the ratio of the convection and the diffusion, called the Peclet number. A good transportiveness implies that there is a good relationship between direction of the flow and the Peclet number. At last the accuracy of the scheme is determined, which indicates the numerical error induced by the discretisation method [Versteeg & Malalasekera, 2007].

Fulfilling all of the above conditions for a second order accuracy, is the class of Total Variation Diminishing schemes (TVD). Although this accuracy is slightly lower than higher order schemes, the outcome is nearly as close to the exact solution. TVD schemes show less false diffusion than the Upwind schemes and do not show any non-physical overshoots and undershoots.

In order to arrive at these results the positive elements of the different schemes are combined by using a weighting towards the upstream cell. This weighting is done with the flux limiter function $\psi(r)$, which is a function of the ratio (r) of the difference with the upstream cell, to the difference with the downstream cell. All usual schemes can be written as a flux limiter, which is plotted in Figure 4-5. This figure, designed by Sweby, can also be used to mark the area of all second order TVD schemes (Figure 4-5) [Sweby, 1984].

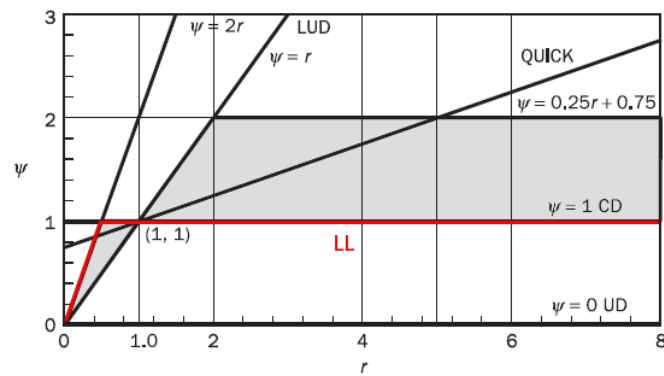


Figure 4-5: Flux limiter functions. The schemes in the figure are standard upwind differencing (UD), linear upwind differencing (LUD), central differencing (CD), the higher order QUICK scheme and OPENFOAM's limitedLinear (LL).

Schemes only in the marked area fulfil all TVD requirements. (Altered from [Versteeg & Malalasekera, 2007])

None of the default discretisation schemes are within the marked area for all values of r , but many limiter functions have been developed that do fulfil all requirements. Within OPENFOAM the limitedLinear is used. With the default value of $k=1$ it is similar to the well known SUPERBEE and Sweby flux limiter functions. The limitedLinear flux-limiter is also shown in Figure 4-5.

The exact coefficients for the above methods within OPENFOAM are given in Appendix F.

4.4 MODEL CALIBRATION

Now that the model has been configured, the agreement to the measurements of Van Doorn can be determined. Before arriving at this complete case, the response of the numerical model to the body force term in the momentum equations is tested. At first this is done for a simple 1D simulation to look at the agreement to the expected flow velocity. In the next paragraph the full implementation of the body force functions is tested, including the hub and the changing body force over the radius. In this paragraph a comparison is made between both body force distributions as shown in Chapter 2.2. Thereafter the full model setup as described in Chapter 4.3 is applied and the model is calibrated to agree with the physical model meas-

urements. The resulting velocities for this simulation are compared to the measurements and to the theoretical solution of the Dutch approach.

4.4.1 FLOW GENERATION IN A SIMPLE 1D SIMULATION

For a simple one dimensional mesh (20 x 1 x 1 cells) the velocity generation of a laminar flow is simulated for a body force added in a single cell. The resulting velocity and pressure are shown in Figure 4-6.

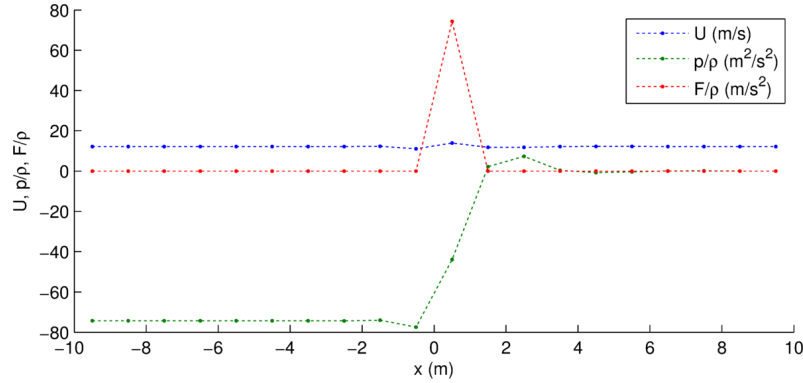


Figure 4-6: Change of pressure, velocity and body force for a one dimensional simulation

It can be seen that the velocity is approximately constant due to the continuity equation applying a constant discharge condition over the domain. The velocity does show a small wiggle to -10% and +10%. A wiggle is also visible in the pressure, which changes over both cell faces of the body force cell from a negative to a zero pressure, due to zero-pressure boundary condition applied at the downstream ($x = 10$ m) side of the domain. This difference in pressure is similar as happening at propellers as explained in Chapter 2.1, with the upstream suction side and the downstream pressure side of the propeller blades. The wiggles are possibly the result of not correcting the pressure equation for the body force implementation as described in 4.2.2.

Integrating the force (F) over the cell and multiplying with the density results in a total force of $74.3 \cdot 10^3$ N. The thrust is calculated with Equation (2.4) and results in a uniform $149 \cdot 10^3$ N over the entire domain. The momentum does not increase due to the constant discharge and the resulting constant velocity. Instead the numerical model shows a pressure jump of $74.3 \cdot 10^3$ N/m². This can be explained by simplifying Equation (4.2) for constant velocity to Equation (4.5).

$$\frac{1}{\rho} \nabla p = \frac{F}{\rho} \quad (4.5)$$

This pressure jumps will result in a velocity by applying Bernoulli's principle for the conservation of energy. Either the boundary condition will apply this law as shown in Equation (4.4) or Equation (4.6) applies for the full domain. In this equation the right hand side is calculated at a location after the body force application and the sudden pressure jump, while the left hand side is outside the domain where the velocities are zero.

$$(p)_{outsideDuct} = \left(\frac{1}{2} \cdot \rho \cdot U^2 + p \right)_{insideDuct} \quad (4.6)$$

This results in a relation between the force and the velocities that has the velocity a factor $\sqrt{2}$ higher than expected in Equation (2.4).

$$U = \sqrt{2 \frac{F \cdot \Delta}{\rho}} \quad (4.7)$$

4.4.2 COMPARISON OF THE SHAPE FUNCTIONS IN A DUCT

Now the implementation of the body force is shown to be working, a comparison is made of the two shape functions as developed in Chapter 2.2 with the uncorrected and uncalibrated coefficients as given in Chapter 3.1.

For this simulation a long duct is used, with the actuator disc and the propeller hub at the early part of the duct. The wall roughness and turbulence as defined in Chapter 4.3 is now included in the model.

In the top figure of Figure 4-7, the velocity distribution induced by both actuator disc is shown at several locations among the duct. Among the duct the thrust, torque and discharge are computed and the change over the duct is shown at the bottom figure.

It can be seen that the actuator discs generates a uniform discharge in the duct that does not change over the length, as is expected from the continuity. The thrust shows an increase around the actuator disc. This is induced by the propeller hub which redirects the flow to the outer edge of the duct. Over the length of the duct the distribution of the axial velocities goes to its equilibrium distribution. When looking at the tangential component, one can see that at the upstream side of the actuator disc the tangential velocities and the resulting calculated torque are still zero. At the location of the actuator disc the velocities increase to a sinusoidal distribution. The torque slowly decreases as it flows through the duct.

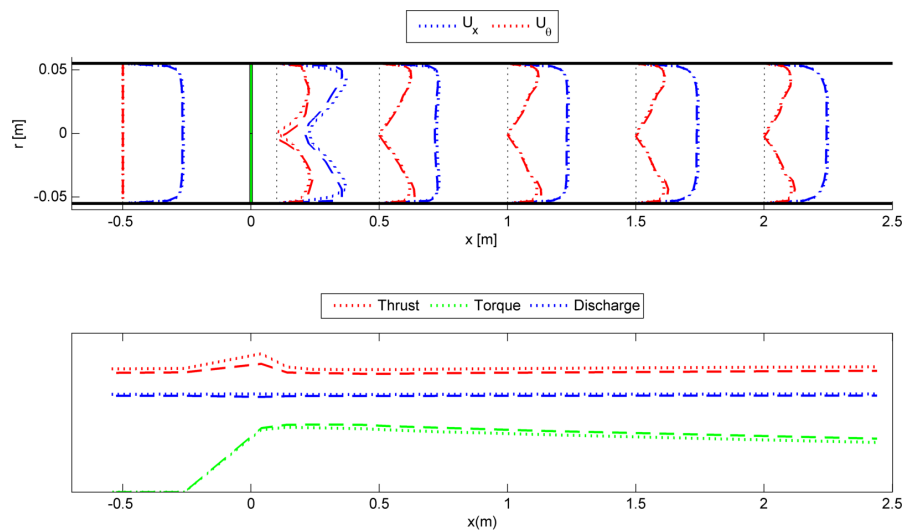


Figure 4-7: Change in velocity distribution (top) and the thrust, torque and discharge (bottom) in the duct.
The Goldstein optimum body force is dotted, the ducted body force is dashed.

The difference in distribution between the *Goldstein optimum* body force and the *ducted* body force in Figure 4-7 is minimal. When looking at the velocity distribution at the pressure side of the actuator disc the difference between both functions and the comparison of the square root of the force distribution to the velocity distribution can be analysed in Figure 4-8.

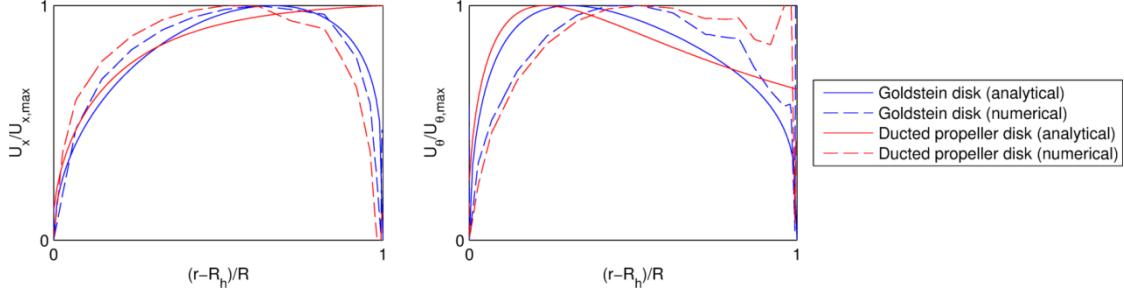


Figure 4-8: Comparison of the analytical to the numerical velocities at the actuator disc

For the axial velocities the shape of the velocities induced by the *Goldstein* actuator disc agree very well with the shape expected from the analytical expression. For the ducted actuator disc the high axial velocities near the outer radius are not showing the numerical model. This is most likely the result of the wall boundary condition which dampens these velocities.

In the right figure the tangential velocities show less similarities to the expected distribution. The maximum has moved toward the middle of the radius. At the duct radius the tangential velocities increase again.

For the further calibration only the Goldstein distribution functions are used, as the resulting effluxes of both functions in Figure 4-7 are very similar. It is used more often and has less arbitrary coefficients.

4.4.3 COEFFICIENT CALIBRATION TO THE EFFLUX

Although the shape of the induced velocities at the propeller might not be exactly as expected it might still be able to calibrate them to the correct shape by changing the coefficients until the efflux as measured by Van Doorn is reproduced. As a starting case the coefficients as derived in Chapter 3.1 are used, which results in the efflux velocities of Figure 4-9.

The coordinates used in x, y and z direction, as defined in Figure 2-4, have their origin at the location where the propeller axis intersects the hull of the ship. This definition is used for all further figures and can be seen in Figure 4-14. For the efflux this means that the location of the measurements is $x = 95 \cdot 10^{-3}$ m, which is approximately equal to $x/D_0 = 0.9$.

In the simulation a disc thickness of $7.5 \cdot 10^{-3}$ m is used. From Chapter 4.4.2 it was concluded that the Goldstein distribution will be used for the simulations. Equation (3.6) and (3.7) can be used to calculate the coefficients for this distribution.

Goldstein optimum coefficients:

$$\begin{aligned} A_x &= 2 \cdot 10^6 \\ A_\theta &= 6 \cdot 10^4 \end{aligned} \tag{4.8}$$

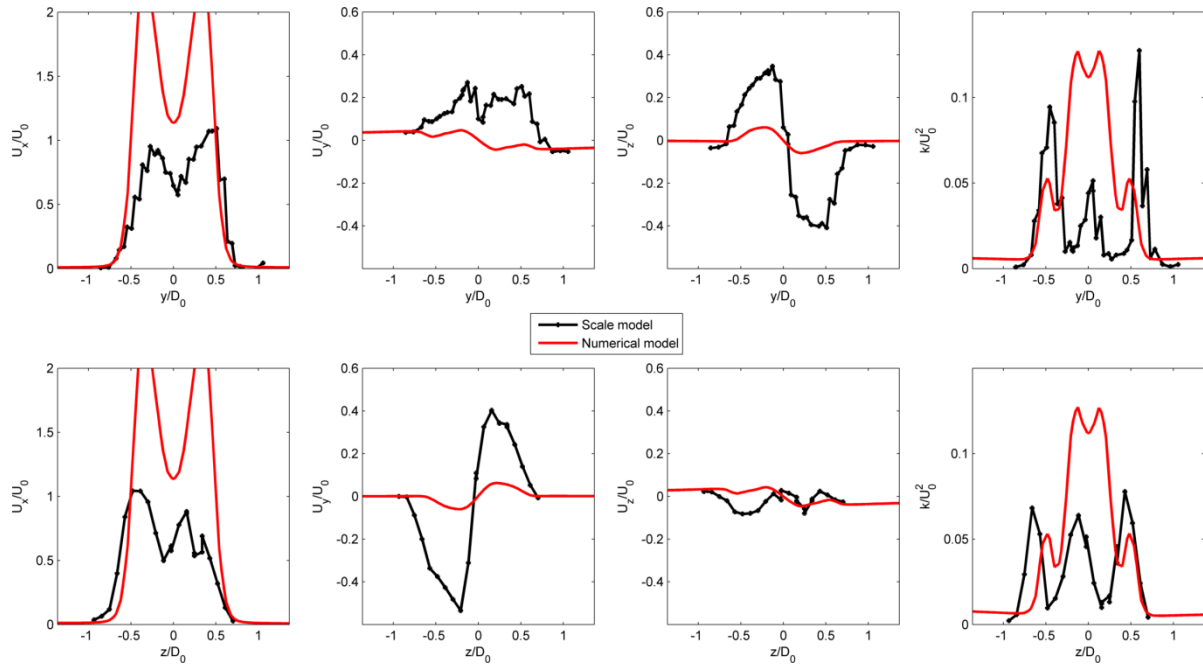


Figure 4-9: Velocities in the water jet of the Goldstein optimum at a distance (x) of 95 mm from the efflux. The plots show the velocities in the horizontal plane (top) and the vertical plane (bottom). From left to right it shows the velocities in axial (x), lateral (y) and vertical (z) direction and the turbulence kinetic energy (k) for both the measurement data of Van Doorn in Scenario 1 and the numerical model with an outflow in a free field with the original coefficients.

The velocities in the numerical model show a large deviation to the measurements of Van Doorn. The axial velocities are higher than expected, while the introduced tangential velocities are nearly negligible.

The coefficients are changed to arrive at a better fit to the efflux. The axial coefficient is reduced, while the tangential coefficient is increased. The obtained relation to the thrust is shown in Figure 4-10. For the thrust this shows a different linear relation, but the torque shows a quadratic relation to the tangential coefficient instead.

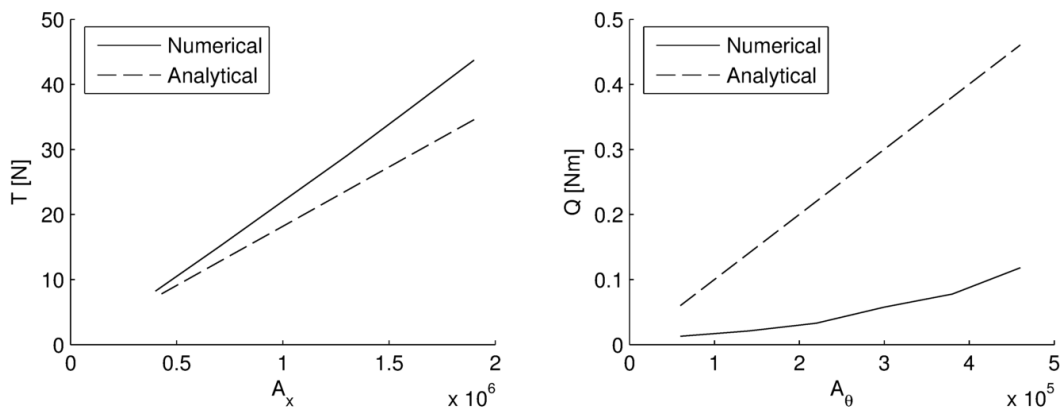


Figure 4-10: Comparison of the relation between the coefficients and the thrust/torque for the Goldstein optimum. The thick line indicates the measured relation and the thin black line is the analytical relationship.

The calibrated efflux is obtained at first by finding the correct thrust and torque value in this figure, but as the derivation of the thrust and torque in Chapter 3.1.2 had a large inaccuracy

due to the scattered data points by Van Doorn, the final calibration is derived by comparing the efflux to the measurements by eye. This results in the coefficients $A_x = 1.0 \cdot 10^6$ and $A_\theta = -3.5 \cdot 10^5$ which is also plotted in Figure 4-11. The tangential component is negative to agree with the correct rotational direction. This high reduction in axial coefficient is expected from the results of Chapter 4.4.1, which showed that the resulting velocities were $\sqrt{2}$ higher than expected from Chapter 2.2.1. Therefore a reduction by this same factor is expected. The rest of the reduction can be explained by the inaccuracy of the calculated thrust in Chapter 3.1.2. The thrust in the numerical model shown in Figure 4-11, is only 24 N. Applying this thrust to Equation (2.10) and multiplying by $\sqrt{2}$ results in a value of $A_x = 0.93 \cdot 10^6$, which is lower than necessary in the numerical model. The increase in axial coefficient that is needed for the correct efflux velocities is needed to compensate for the energy losses in the duct due to wall resistance.

The tangential component is increased by a factor 6 to achieve a fit to the measured tangential velocities in the efflux with a torque of 0.11 Nm. This large increase might be the result of the circulation induced by the pressure difference over the propeller radius, which will result in diffusion of those velocities, but it is also possible that the difference is partly induced by a similar theoretical error, as the expected linear relation between the coefficient and the torque in Figure 4-10 is not linear but quadratic. The possibility for this type of error has not been tested.

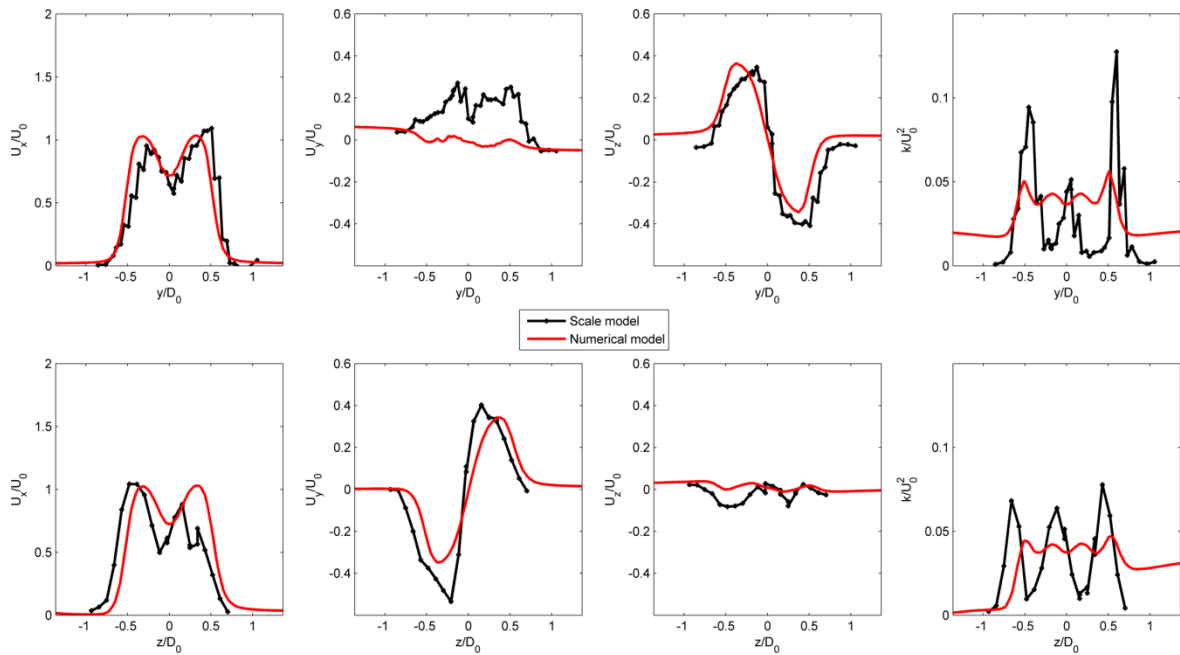


Figure 4-11: Outflow of the calibrated numerical model.

Although the numerical model is an accurate approximation, it still shows some deviations. As the outflow of the numerical model is symmetric, it does not show the asymmetry of the scale model measurements, although these measurements might also possibly be inaccurate as discussed in Chapter 3.1.1. The worst fit is found in the plot of the turbulence kinetic energy at the right of the figure. Although the order of magnitude is similar, the numerical model shows

a nearly uniform distribution, where the physical model measurements show large peaks at the location of the propeller tips and at the propeller hub. This is expectable as extra turbulence, induced by amongst others cavitation, is not included at the actuator disc.

4.4.4 DIFFUSION OF THE WATER JET

The diffusion of the water jet generated with the calibrated actuator disc can be compared to more measurement results and to the analytical diffusion of Equation (2.21). For the analytical diffusion the uniform efflux velocity is calculated with Equation (3.4) for a radius of 0.055, resulting in $U_0 = 1.59$ m/s.

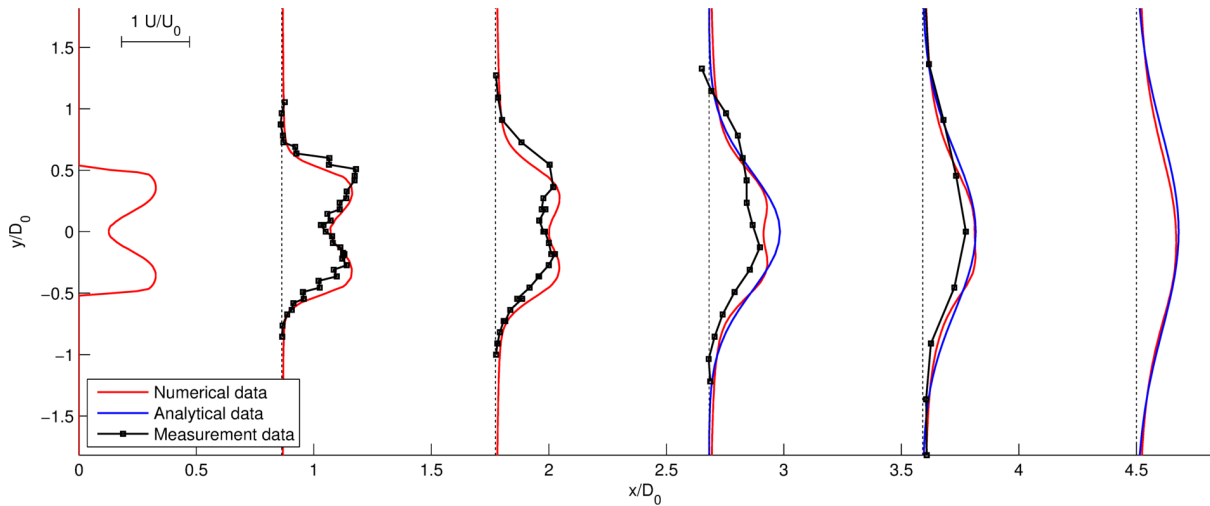


Figure 4-12: Diffusion of the water jet as a function of the dimensionless distance to the efflux at $z/D_0 = 0$.

As the measurement data and numerical data were calibrated to $x/D_0 = 0.9$ they show good comparison at this location. At $x/D_0 = 1.8$ the shape and maxima of the velocity are still equal, but the measurement data show a shift in positive y -direction. This is probably due to an error in the set-up where the propeller has been slightly rotated towards this direction. This effect is also noticeable at the farther locations, but also more diffusion can be noticed at these locations. In the numerical model the location of the low-velocity core has only fully disappeared at $x/D_0 = 4.5$, which is later than expected in the (less accurate) earlier measurements of Chapter 3.1.

The analytical solution of Blaauw and van der Kaa is included in the figure for $x/D > 2.5$, where the low velocity core of the efflux has mostly disappeared. It shows a very good agreement with the numerical model which proves the validity of the analytical solution for an unobstructed outflow.

4.4.5 VELOCITIES AT A SLOPE COMPARED TO THEORY

As no measurements were done exactly in line with the outflow of the propeller a comparison at this location cannot be made between the numerical model and the measurements at that exact point. Instead just the comparison is done with the diffusion with the analytical diffusion of Blaauw and van der Kaa as shown in Chapter 2.3.3. This is shown in Figure 4-13.

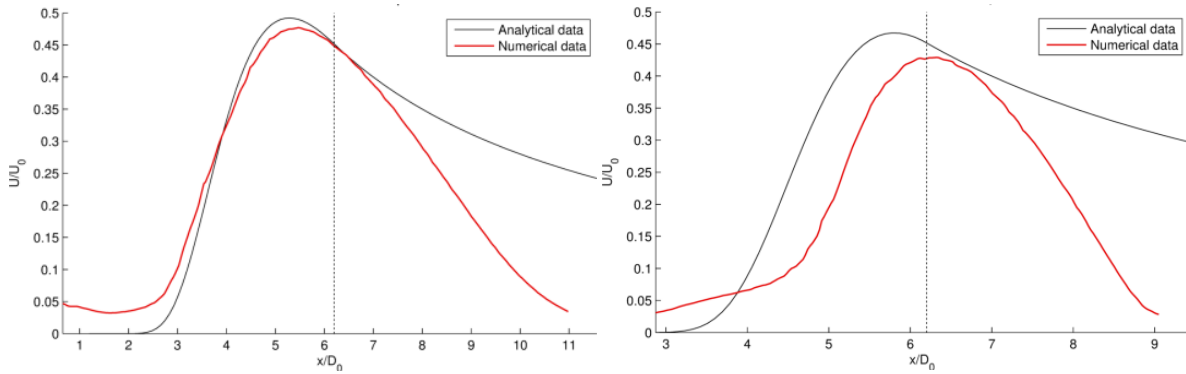


Figure 4-13: Total velocities on the slope for the theory of Blaauw and van der Kaa and for numerical simulations over an axial line at $y/D_0 = 0$. At the left it shows a slope of 1:2.5, at the right it shows a slope of 1:1.5. The dotted line indicates location of intersection of the propeller axis with the slope.

The comparison shows that both the location and magnitude of the velocities show a good agreement for the 1:2.5 slope. At higher locations on the slope the analytical model shows higher velocities, but it was already noted in Chapter 2.3.3 that the formula is not valid at those regions.

For the steeper 1:1.5 slope there is a significant difference between the analytical and numerical solution. The theory has a higher maximum slope velocity, which also occurs at a location beneath the propeller axis, while the numerical model has its maximum at the intersection of the propeller axis and the slope. A comparison to the measurements to similar locations is necessary to validate which model gives the best approximation.

4.4.6 VELOCITIES AT A SLOPE COMPARED TO MEASUREMENTS

With the calibrated water jet the velocities on a slope are computed. Of the many cases of Van Doorn, three geometries are used for comparison between the measurements and the numerical model. These three cases are Scenario 1 (a smooth slope of 1:2.5), Scenario 2 (a smooth slope of 1:1.5) and Scenario 6 (a smooth slope of 1:1.5 with piles). The full set of figures, also including some other scenarios, can be found in Appendix G.

SCENARIO 1

The first scenario is a basic scenario with a relatively gentle slope. Measurements by Van Doorn were done in an axial line located nearly in line with the efflux and at three lateral lines at a high location on the slope (left of Figure 4-14).

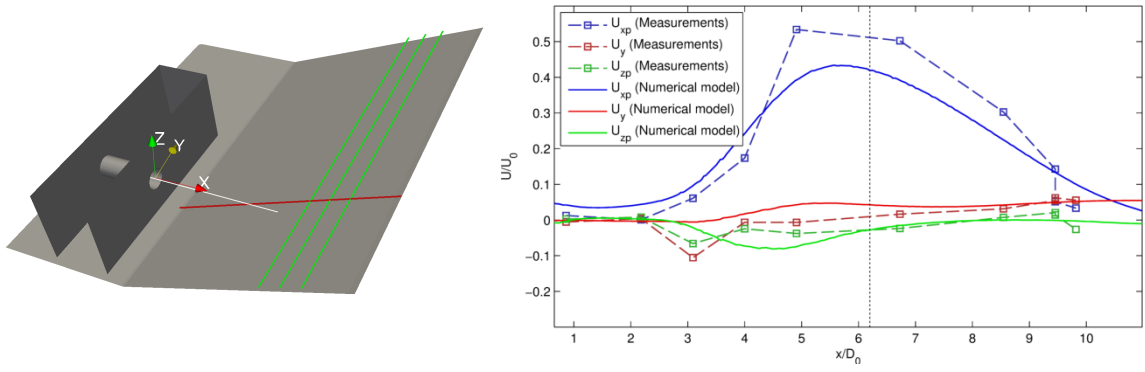


Figure 4-14: (left) The locations of the axial (red lines) and lateral (green lines) plots of Scenario 1; (right) Velocities at the slope over the axial distance at $y/D_0 = 0.4$. The dotted line is the intersection of the thruster axis and the slope.

For both the calibration in axial direction (right of Figure 4-14) and the calibration to the lateral direction (Figure 4-15) the numerical results show a good fit to the scale model measurements of Van Doorn. The measurements show a very high deviation from point to point. This is probably the result of incorrectly registered measurements by Van Doorn. As leaving out certain measurement points would be arbitrary, the general trend of the correctly-looking measurements is used for the calibration. To these measurements the deviations are analysed.

The shape of the velocities agrees very well. The parallel axial velocities are higher and the lateral velocities in lateral direction reduce more in the physical model. The turbulence kinetic energy at the slope (not plotted) does not appear to have any agreement between both models, this is probably the result of the averaging RANS solving method and the related wall functions, which are not meant for the creation of a good approach of the turbulence at those locations.

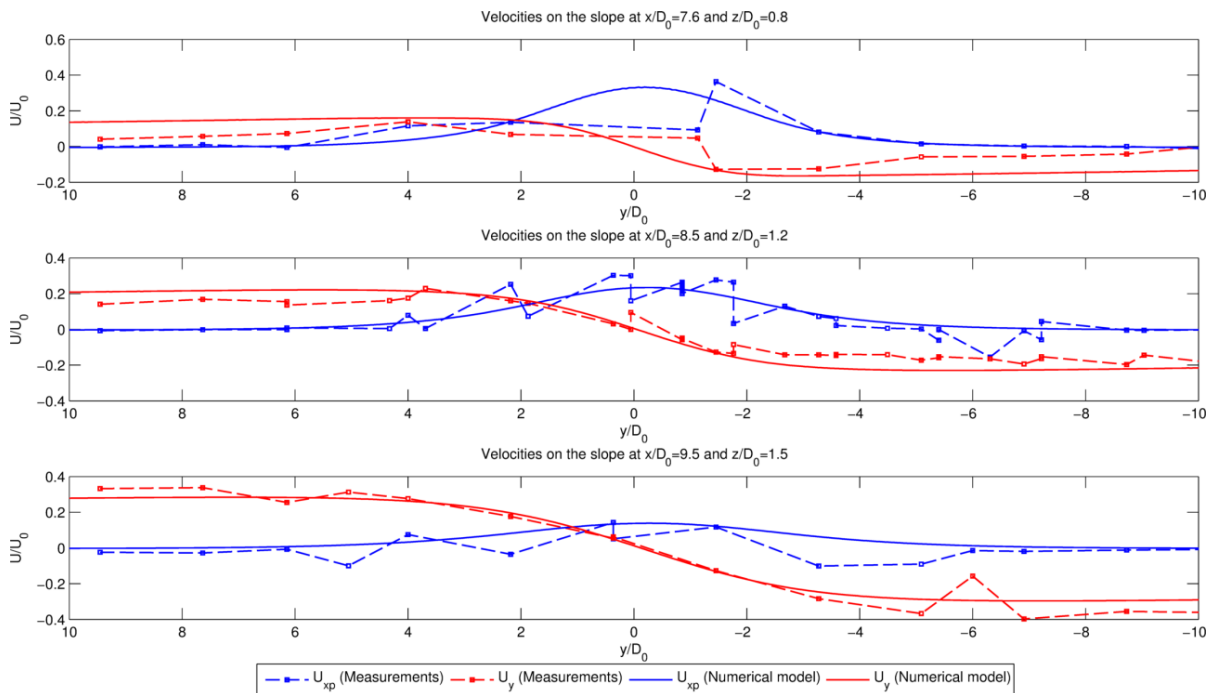


Figure 4-15: Velocities at the slope over the lateral lines. The velocities perpendicular to the slope are not plotted as they are nearly negligible in both the measurements and the numerical model.

SCENARIO 2

In Van Doorn's second scenario a steeper slope was used and the calibration is done to lines on the slope in the lateral direction. Of the 7 lines shown in Figure 4-16, the first, fourth and sixth line from the toe of the slope are shown in Figure 4-17. In Appendix G all seven figures are shown.

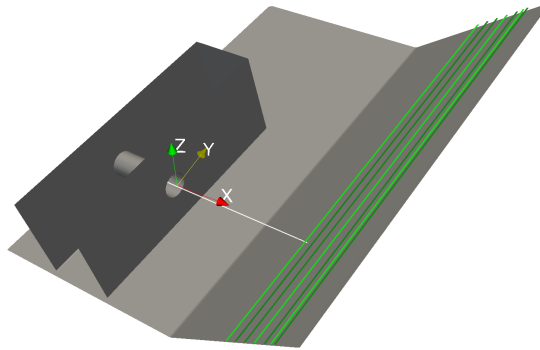


Figure 4-16: The locations of the lateral (green lines) plots of Scenario 2

Similar as for the previous case, the parallel axial velocities in the numerical model are lower than measured by Van Doorn. But in contrast to Scenario 1, the lateral velocities are no longer decaying slower in the numerical model, but show a good comparison. The lateral velocities at the furthest lateral point on the slope are even lower in the numerical model.

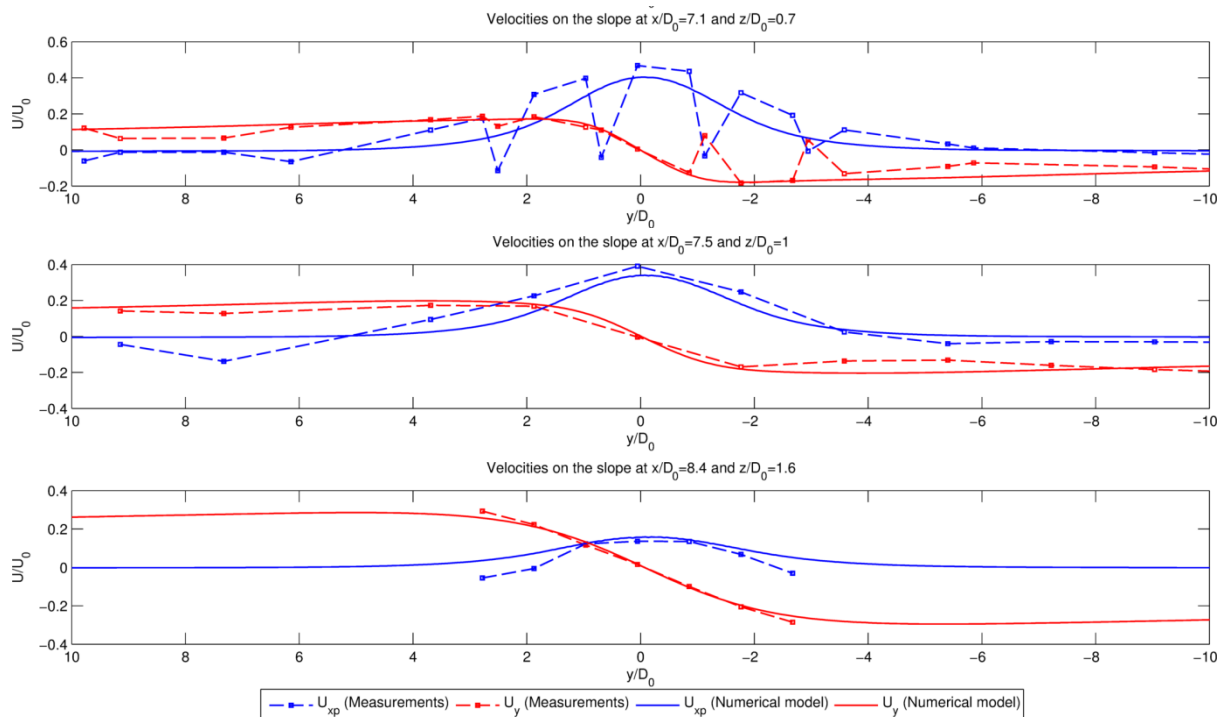


Figure 4-17: Velocities at the slope over the lateral lines

SCENARIO 6

The sixth scenario of Van Doorn introduces piles to the second scenario. In contrast to previous cases, measurements were now also done at the lower regions of the slope (left of Figure 4-18). Comparison over the second, fourth, seventh and eighth line from the toe of the slope is shown in Figure 4-19 and over the axial line in the right of Figure 4-18, which is located at $y/D_0 = 0.7$.

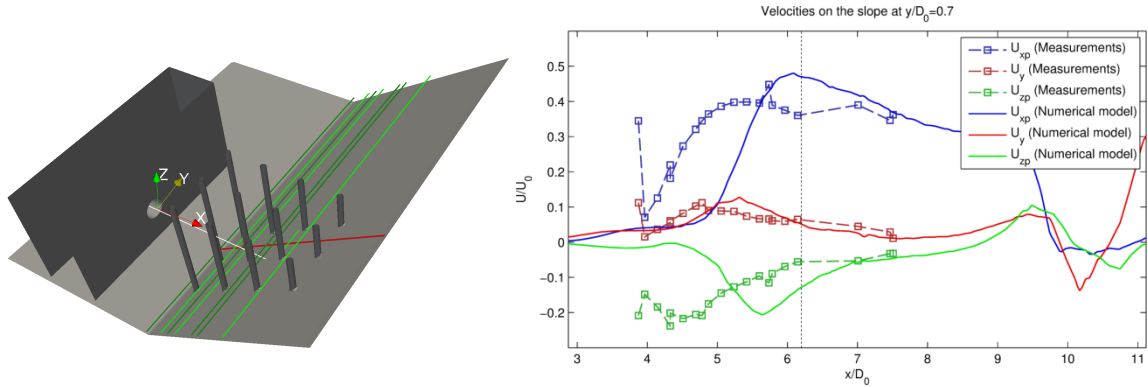


Figure 4-18: (left) The locations of the axial (red lines) and lateral (green lines) plots of Scenario 6; (right) Velocities at the slope over the axial distance

In these measurements it shows that although the maximum velocities on the axial line show a very well agreement between the physical and numerical model, the velocities at the lower part of the slope are significantly lower in the numerical model than the physical model. Apparently the radial spreading in the numerical model for steeper slopes is too low to capture these velocities. This did appear in the simulation with smoother slopes (Figure 4-14) and might be caused by an error in OPENFOAM.

At the bottom of the slope a large deviation between the physical and numerical scale model is visible. Where the velocities in the numerical model have mostly disappeared, the measurements of Van Doorn still show strong velocities in both parallel and perpendicular direction. It is concluded that the radial spreading of the water jet is underestimated in the numerical model. This agrees with the conclusion of Chapter 4.4.5, where the theoretical water jet also showed higher velocities on lower regions of the slope.

A possible reason for this effect occurring in numerical models with the steeper (1:1.5) slope, might be the implemented wall boundary layer. This could have resulted in a wall roughness in the numerical model that is higher than the (smooth) roughness in the measured scenarios. A higher roughness can subsequently lead to lower velocities of the water jet.

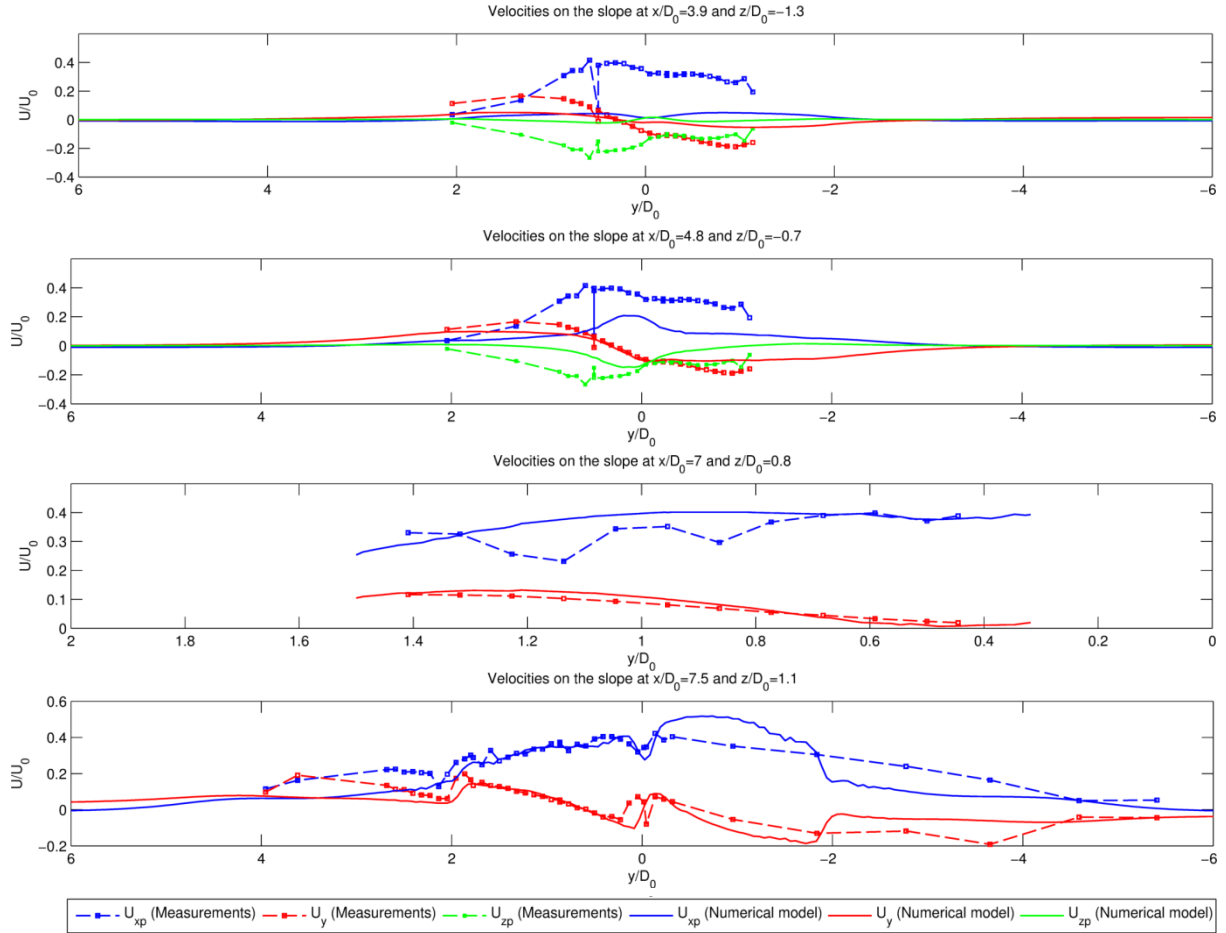


Figure 4-19: Velocities at the slope over the lateral lines. The third figure at $x/D_0 = 7$ is located between two piles the range of the y/D_0 is changed.

4.5 CONCLUSION - USING THE MODEL

A numerical scale model has been set-up that approximates both the theoretical velocities and the measurements in the scale model of Van Doorn reasonably well. For the set-up of the model a domain with open boundaries is used, to neglect undesirable influences of the domain (or basin) edges. The turbulence is modelled by using the Realisable k-epsilon RANS model, which is able to accurately simulate swirling water jets for low computational costs.

The model does not include the geometry of the propeller itself, as this would result in a computational expensive velocity field that could not be further calibrated. Instead the propeller velocities are approximated with an actuator disc which locally adds a body force to the momentum equations. A local increase of the turbulence at the hub and the propeller tip is not implemented in the numerical computations.

In the calibration phase of the model it was concluded that the relation of a uniform body force to the resulting efflux was different than expected in Chapter 2.2.1. A difference of $\sqrt{2}$ was noticed in the simple numerical model and an explanation was given by applying Bernoulli's law to a duct with constant momentum.

Comparing the body force distribution of the open and ducted propeller showed a minimal difference in the resulting velocities. The velocity distributions did, however, differ from the shape expected for the body force functions. When comparing the efflux of the *Goldstein* distribution function to the measurements of Van Doorn, calibration is needed and the coefficients need to be updated. For the axial coefficient a small increase is needed to compensate for the losses due to wall resistance, but the tangential coefficients need a large amplification to correspond with the desirable torque.

The calibrated efflux is compared to both theoretical and measured velocities at different locations. Comparing the diffusion of the water jet to the theory of Blaauw and van der Kaa shows an exact agreement, while the measurements of Van Doorn show more and asymmetric radial diffusion.

At the slope the theory of Blaauw van der Kaa also shows a good fit to the numerical model for a gentle slope of 1:2.5 near the intersection of the propeller axis and the slope. At location higher and lower on the slope the theory is no longer valid and deviations are notable. For a steeper slope of 1:1.5 the numerical model computes a different location and magnitude of the maximum velocities. This same error in the numerical model is shown when comparing it to the scale model measurements, where also high velocities were measured near the toe of the slope. It is suggested that this error might be the result of an incorrect simulation of the roughness of the slope. A high roughness might have resulted in a faster dissipation of the kinetic energy in the water jet near the slope.

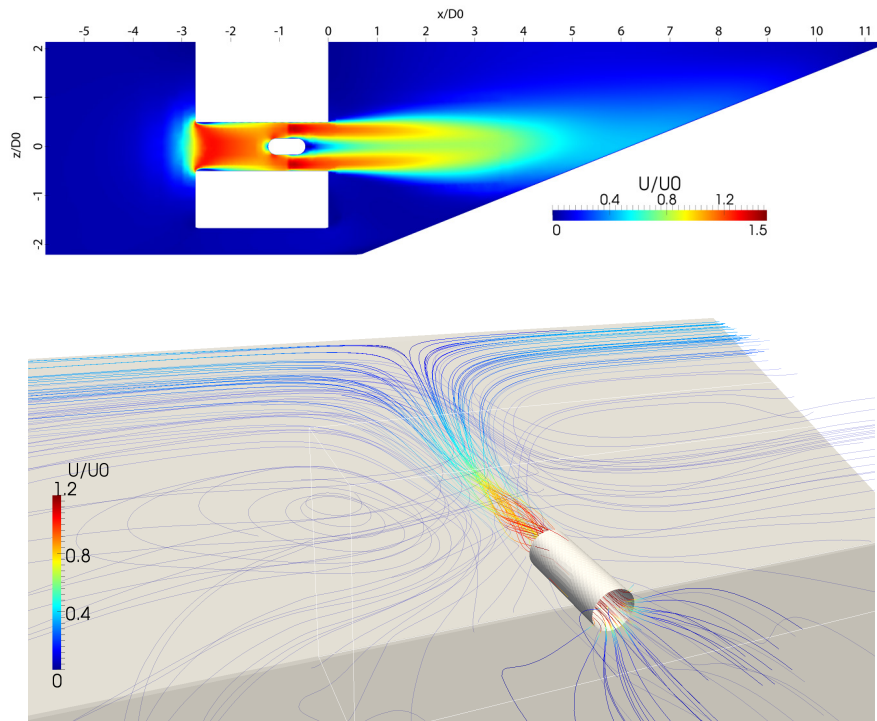


Figure 4-20: Two illustrations showing the water jet of the calibrated RANS model in Scenario 1 of the scale model of Van Doorn. In the top figure the a plane at $y/D_0 = 0$ is shown. It shows the diffusion of the water and the disappearance of the low-velocity core. In the bottom figure a streamline plot shows the rotation of the water jet and the spreading of the velocities over the slope. For particles that did not pass through the bow thruster, the opacity is reduced. In white the outline of the ship is shown.

5 SENSITIVITY ANALYSIS

Although the model shows good results when compared to the measurements, it is questioned how stable the resulting efflux is for small changes in and around the actuator disc. In *Chapter 5.1* the different parameters of the actuator disc are changed and compared as well as the mesh around the actuator disc.

Besides testing the actuator disc the model is also tested for changes in the mesh (*Chapter 5.2*) and roughness functions at the slope (*Chapter 5.3*). Different turbulence models are compared to compare the influence on the results, in this process also a transient LES simulation was performed.

In the last sensitivity analysis, the body force coefficients are changed and for the different efflux velocities the dimensionless velocities are computed (*Chapter 5.4*). According to theory no changes should be observed.

5.1 CHANGING THE ACTUATOR DISC

At first the parameters of the actuator disc are subjected to the sensitivity analysis. The thickness is checked to confirm that it has no influence on the efflux and the location of the thruster is changed to see its influence on the efflux. At last a sensitivity of the velocities on the slope is done for cases where the tangential component is increased or neglected, or when a radial component is added to the actuator disc.

5.1.1 THICKNESS DISC

The thickness of the actuator disc is reduced to a thickness of one cell ($4.0 \cdot 10^{-3}$ m) and to double, triple and quadruple thickness of the calibration case value ($7.5 \cdot 10^{-3}$ m) and compared to this basis value. It is expected that this has no influence on the results, but the wiggles of the Rhie-Chow interpolation might give an error for sudden body forces, as discussed in *Chapter 4.1*. The comparison is shown in *Figure 5-1*.

All thicknesses show a nearly equal velocity profile. The small deviation that does exist is probably the result of a small deviation of the analytical actuator disc volume to the numerical actuator disc volume as discussed in *Chapter 4.2.2*. The turbulence kinetic energy shows a higher deviation, but this is also insignificant.

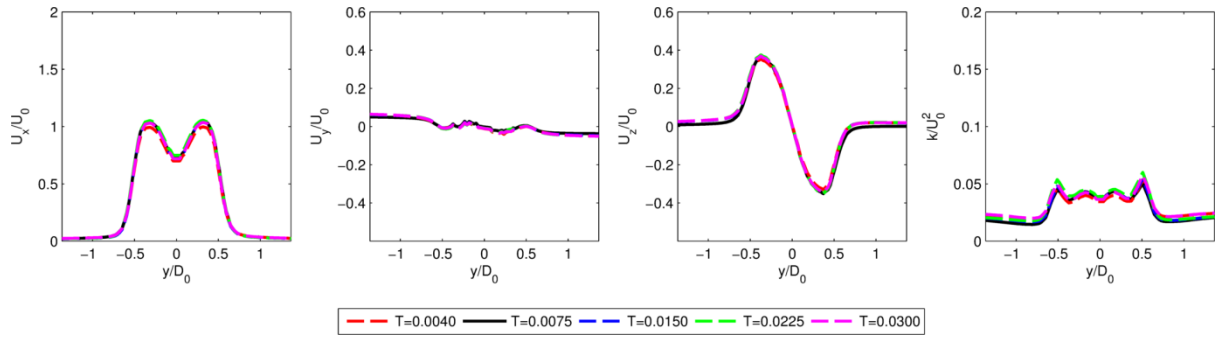


Figure 5-1: Efflux velocities for different disc thicknesses

5.1.2 LOCATION ACTUATOR DISC

The location of the actuator disc in the basic model to the centre of the duct ($L = 60 \cdot 10^{-3}$ m) is based on an early drawing of Van Doorn as was written in Chapter 4.2.2. As this makes the location within the duct an uncertain property, the efflux is compared to other location in the duct in Figure 5-2. It was moved to the back of the duct, to the centre and to the front of the duct. For the front actuator disc location the hub was already partly outside the duct.

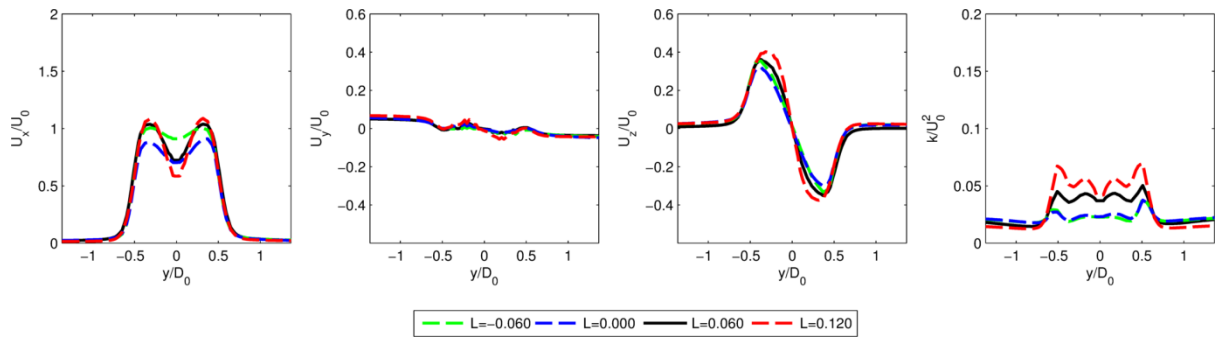


Figure 5-2: Efflux velocities for different locations of the propeller

It can be seen that the further the actuator disc is located to the front, the deeper the low-velocity core in the water jet, the steeper the curve in the tangential direction and the higher the turbulence kinetic energy in efflux. This can all be attributed to the reduced distance to the measured location and thereby a reduction in the diffusion from both duct and the viscosity. A reason for the lower total thrust and torque for the centred disc location is not found as the rest of the scenario is exactly identical.

The situation with the propeller located at the front of the duct, might improve the results as the lower core and higher turbulence energy are also shown in the measurements. But as with certainty can be said that the propeller could have never been placed at that location and it could only have been placed further to the back of the duct, no changes were done to the model.

5.1.3 ADDING RADIAL VELOCITIES AT THE ACTUATOR DISC

In Chapter 4.4.3 it was shown that the radial spreading (or diffusion) of the water jet in the numerical model was less than measured in the physical model. To increase the spreading of the water jet an arbitrary radial body force is added to the actuator disc. Equation (5.1) shows

the implementation of the radial component, in which it is made a function of the tangential component and thereby has the same shape with a different direction.

$$f_r = f_\theta \cdot R \quad (5.1)$$

In Figure 5-3 some of the runs are shown with arbitrary values of the coefficient R. It can be seen that the radial velocities have not at all increased, but instead have resulted in a larger low-velocity core and a great increase in turbulence. This can only be noticed in the R=10 case, as the effect of the radial component for a body force in the same order of magnitude as the tangential component does not have any influence on the efflux.

The high turbulence generated in the R=10 case results in a high dissipation of the efflux velocity. Apparently the adding of radial body forces does not help the radial spreading.

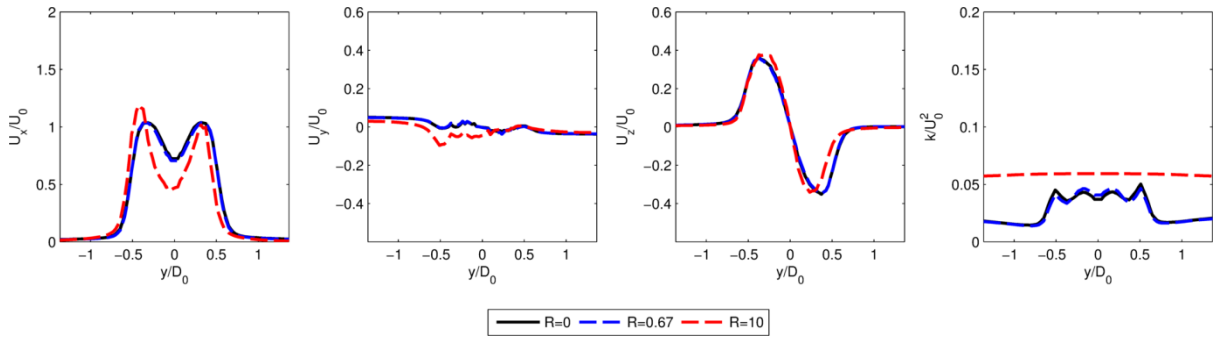


Figure 5-3: Efflux velocities for different radial coefficients

5.1.4 INFLUENCE OF THE TANGENTIAL COMPONENT

For the modelling of a propeller the simulation by a water jet was suggested in which case no tangential velocities would have been apparent in the outflow [De Jong, 2003]. Although tangential velocities were added in later simulations, the effect on the slope was not described.

To predict the difference in velocities on the slope additional cases were run in the piled scenario (Scenario 6) for no tangential velocity and for double tangential velocity as shown in Figure 5-4. Besides the expected increase in tangential velocities, there is also a small deviation in the axial velocities as a slightly larger low-velocity core appears for higher values of A_θ . It also results in a higher turbulence kinetic energy.

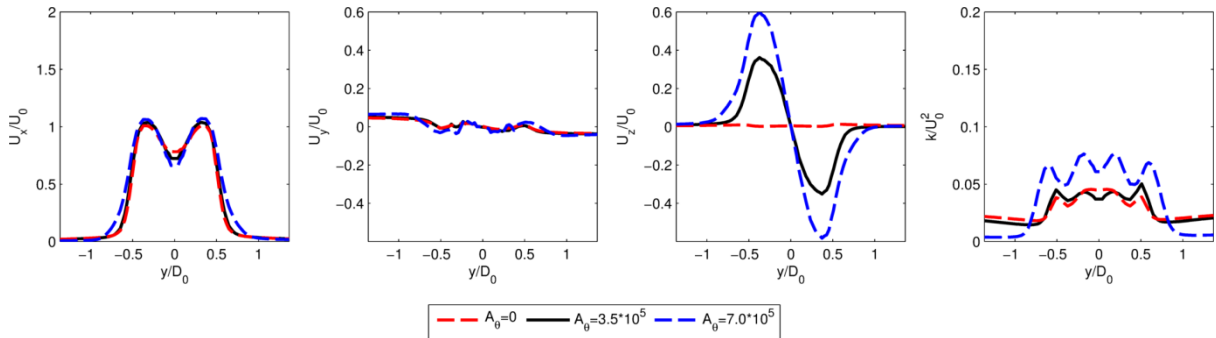


Figure 5-4: Efflux velocities for different tangential coefficients

For the piled slope of Scenario 6, this resulted in a velocity distribution at the slope as shown in Figure 5-5. The tangential velocities give more dissipation and lower maximum parallel velocities. However, the anti-clockwise rotation results in a different flow direction behind the pile as shown in the right figure. The low-velocity trough behind the pile is clearly visible, but this results in lower velocities to the right and an increase in the velocities to the left for the high tangential velocity simulation. When no tangential velocities are added, the water jet has its lateral velocities at a higher location on the slope and thus shows a reduction of the lateral velocities in the middle region shown in the figure.

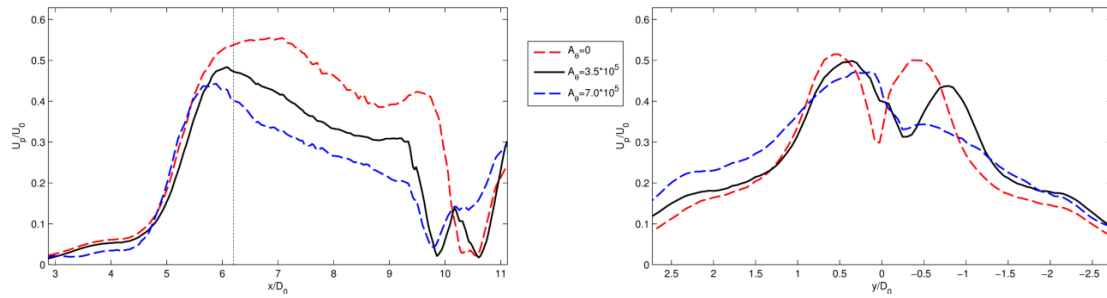


Figure 5-5: Velocities at the slope for different tangential coefficients in axial direction at $y/D_0 = 0.7$ (left) and lateral direction at $z/D_0 = 0$ (right).

5.2 CHANGES TO THE MESH

The quality of the mesh is of great importance for an accurate numerical model. Especially near structures the distance to the nearest cell centre is of importance for a correct modelling of the wall boundary layer. The influence of the layers and refinements to the velocities is verified for both the duct and the slope. Also a comparison is made to the scenario where not the bottom nor the surface has any influence on the now unconfined water jet.

5.2.1 MESH OF THE PROPELLER DUCT

Different thruster meshes were tested for its influence on the efflux of the thruster. Changes were done to the near-wall layers and to the refinement level of the duct. This resulted in the meshes as shown in Figure 5-6. The figure shows for each of them the range of non-dimensional wall distances values which were computed for the full duct. Although none of the meshes shows an y^+ -value far outside the preferable range, the velocities might already be influenced to some extent.

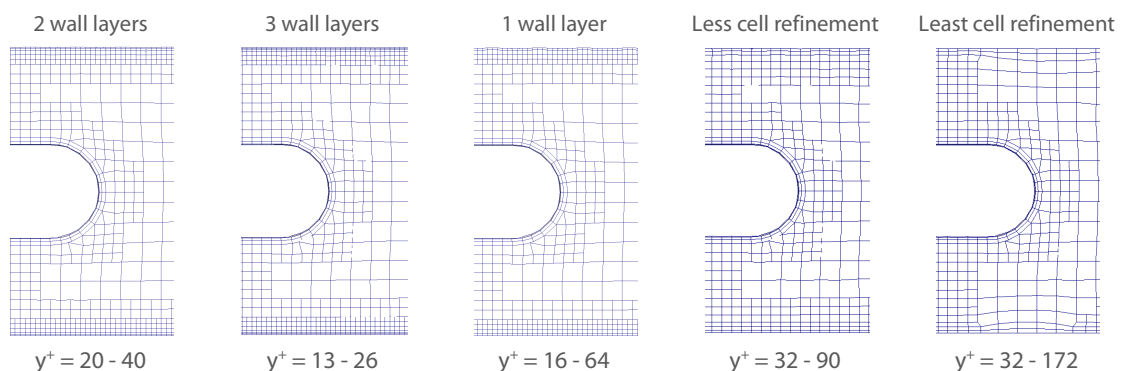


Figure 5-6: Meshing within the duct with the resulting y^+ value. The left mesh is used in the calibrated model

In Figure 5-7 the resulting effluxes are shown. Apart from a slightly lower turbulence kinetic energy for the least cell refinement no differences appear and the actuator disc can be regarded stable for different meshes.

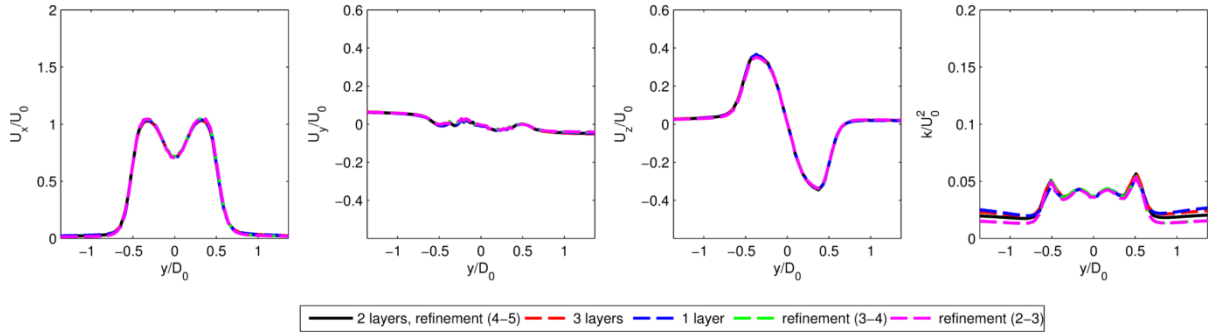


Figure 5-7: Efflux velocities for duct meshes

5.2.2 MESH AT THE SLOPE

A similar comparison is done to the layers at the slope for the gentle slope with no piles. The calibrated (four layers) mesh is compared to grids with more layers (six and five layers) and grids with less wall layers (three, two and one layer). By changing the number of wall layers the non-dimensional wall distance changes as shown in Figure 5-8. The changes in the grid due to the changes in wall layers have a negligible influence on the rest of the grid as the layers are very thin compared to the (refined) grid cells in the inner domain (as can be seen in Figure 4-4 for 4 layers).

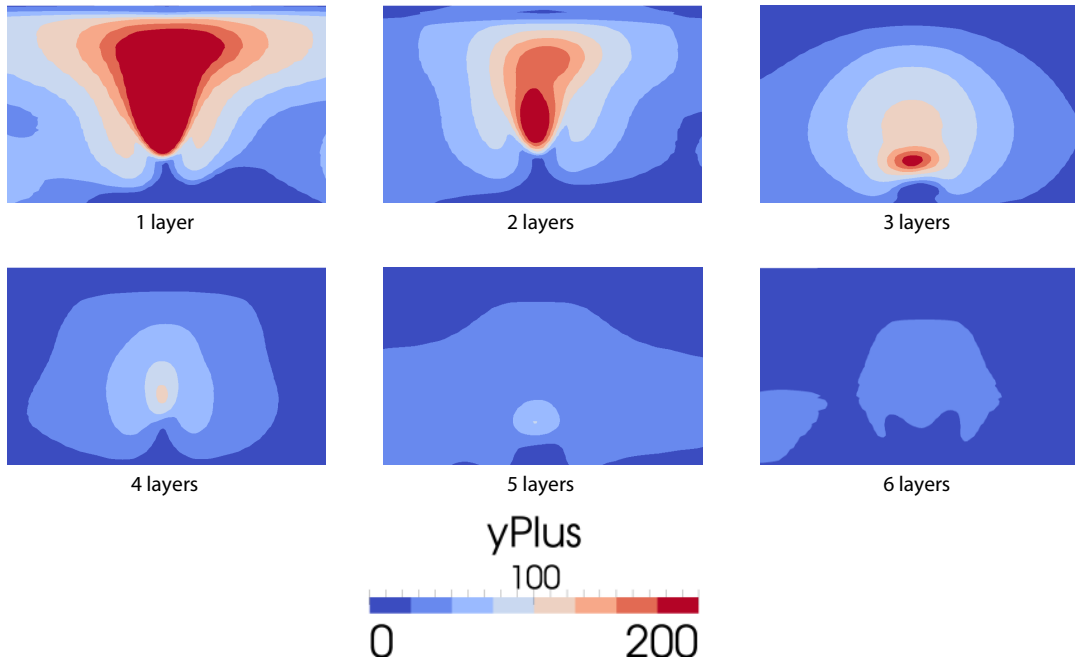


Figure 5-8: The non-dimension wall distance (y^+) for a changing number of wall layers in the mesh. The plots show the full height of the slope, for a width of $x/D_0 = 18$. The situation with 4 layers is the basis mesh. The propeller axis intersects the slope at the centre of these plots.

As earlier concluded in Chapter 4.3.2 the wall functions should show the best behaviour for wall distances between 30 and 100. At the location of highest flow velocities, around the cen-

tre of the figure, the y^+ are too high for the meshes with the least layers, but it should show similar results for 4 to 6 layers. In Figure 5-9 the six layers are shown and supplemented, by a four layer scenario where instead the boundary condition of the slope is changed from a no-slip to a slip condition. Which means that now only for the perpendicular velocities the $U_{zp} = 0$ boundary condition applies.

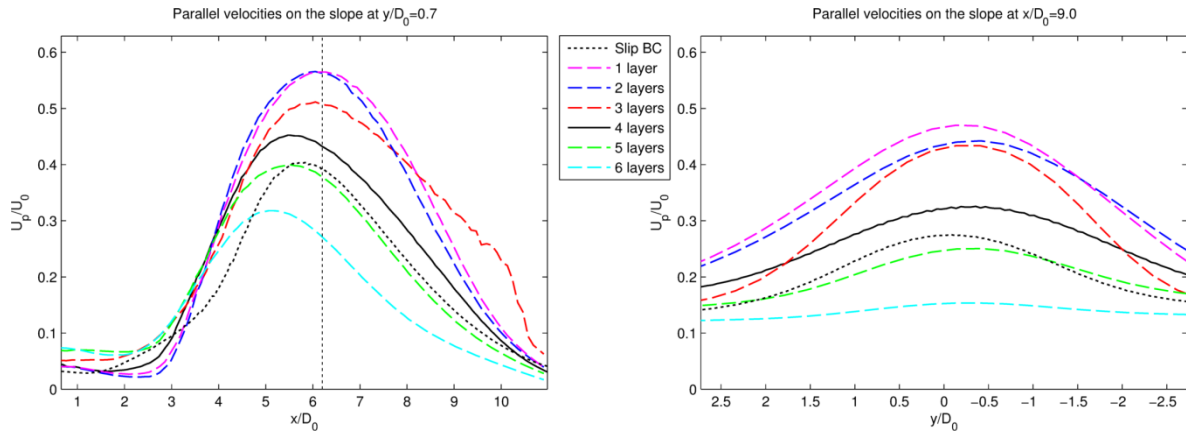


Figure 5-9: Influence on the number of layers near the wall, and thereby different wall distances, to the velocities at the slope in axial direction (left) and lateral direction (right). Also the 'Slip BC' indicates the velocities for the grid with 4 layers when instead of a no-slip wall boundary condition, a slip boundary condition is used at the slope

At first the no-slip boundary condition ('4 layers') is compared to the slip boundary condition. This shows a remarkable effect, where the slip boundary condition shows lower flow velocities than are computed in the no-slip situation. As the flow velocities are allowed to have parallel velocities in the slip-situation an increase in velocities was expected.

Furthermore a relation between the number of layers and the flow velocities can be seen. An increase in layers, resulting in a decrease in non-dimensional wall distance, will result in a decrease in velocity. Although it was expected that all meshes that satisfied the y^+ range should show similar results, all results show a large difference and do not converge to one solution. Only the two and one layer flow plot show similar results, but should both show inaccurate results.

An odd velocity distribution can be seen in the 3 layer case. In this computation the velocity location appeared in line with the other runs, but higher velocities are computed at the top of the slope, while lower velocities are found in lateral direction.

5.2.3 CONFINED OR UNCONFINED JET

The efflux of the velocity should not experience influence of the surrounding geometry. However, as the calibration measurements were done at a distance from the efflux, some influence might be noticeable. In Figure 5-10 the comparison of the outflow over a slope is compared to an entirely free outflow. The turbulence in the latter case is lower than in the case of a water jet over a slope, but also the predominant direction of the z -velocities changes from a positive to a negative value. This is probably induced by the water taking the easiest route, which is

under the ship in the case of an open outflow with no bottom included. It can be concluded that, as expected, the geometry of the surrounding scenario has no influence on the efflux of the bow thruster.

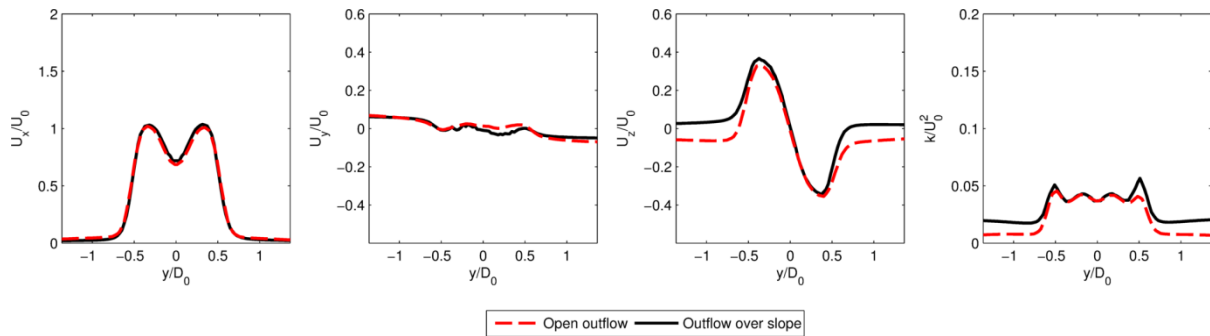


Figure 5-10: Efflux velocities for open outflow and outflow over a slope

5.3 WALL FUNCTION ROUGHNESS

As discussed in Chapter 4.3.3 the default wall boundary conditions of OPENFOAM do not allow for the adjustment of the roughness of the walls. For the turbulence kinetic energy and dissipation rate only the previously applied wall boundary can be applied, but changes are done to the different boundary layers for the turbulent viscosity.

In Table 5-1 the compared wall functions are shown. Choice for the coefficients is done based on advise of the community, but no official recommendations exist for the coefficients or the background of the wall functions. Most rough wall functions are based on other CFD packages, but in depth research of the coefficients was not done. In this table Ks stand for the sand-grain roughness height (0 for smooth walls) and Cs is a roughness constant in the range of 0.5 - 1.0, but often chosen as 0.5.

Table 5-1: Wall functions for the turbulent viscosity

	Type	Coefficients
R0	nutkWallFunction	-
R1	nutUWallFunction	-
R2	nutkRoughWallFunction	Ks = 0.005 Cs = 0.5
R3	nutkRoughWallFunction	Ks = 0.24 Cs = 0.5
R4	nutURoughWallFunction	roughnessHeight = 1e-5 roughnessConstant = 0.5 roughnessFactor = 1

The velocity profile in vertical direction is show in Figure 5-11 for five location. For one location the velocities can be compared to measurement data of Van Doorn. At this point most models agree and show good agreement with measurement. Only case R3 shows a velocity far below the other models. In this simulation the boundary layer is too high and is even influencing the turbulence kinetic energy at the efflux of the propeller. A change in the bottom and

slope wall function should not have major influence on the efflux, but apparently the viscous wall layer of this model results in a very high dissipation reaching far into the domain.

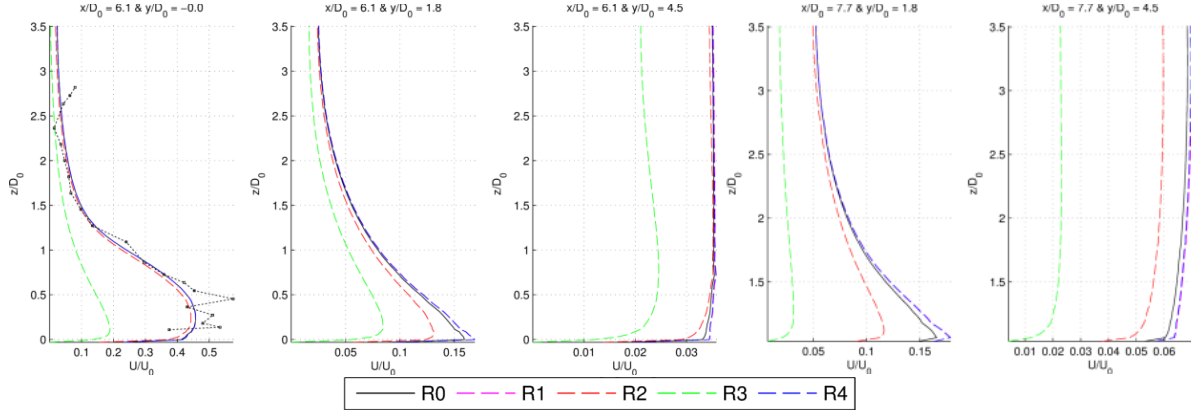


Figure 5-11: Parallel slope velocities at five locations on the slope: directly in front of the thrust and at two locations at two different heights at $y=0.2$ and $y=0.5$ m. The left plot also shows measurement data of Van Doorn (slope 1:1.5).

Comparison to the other models show more comparable results. *R1* is nearly equal to the basis wall function and only shows minimal higher and lower velocities. The same conclusions can be drawn for *R4*, as apparently the roughness height coefficient was chosen too lower to show a wall of even minimal roughness. *R2* shows lower velocities, but also shows a velocity shape showing with a more smoothed boundary layer.

However, besides a better boundary layer, also more dissipation of the velocities occurs (see Figure 5-12), further reducing the velocities at the slope and increasing the deviation to the measurements. Additional research is needed to validate the use of any of the wall functions.

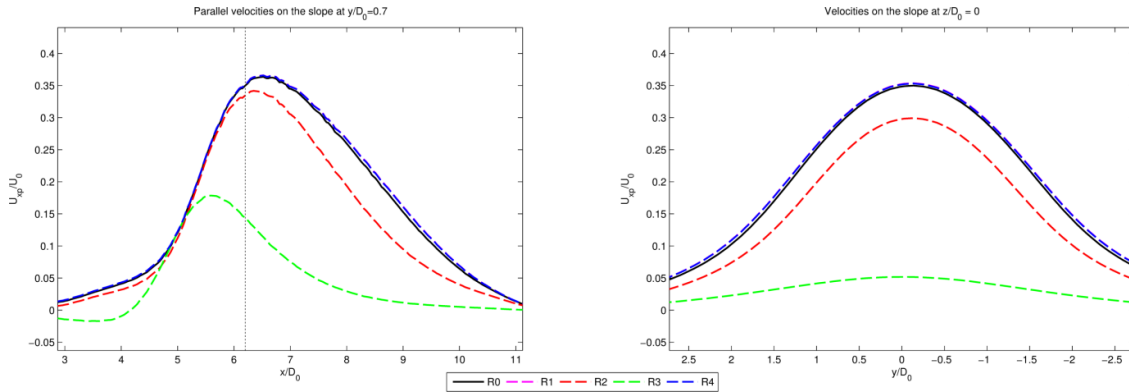


Figure 5-12: Parallel velocities on the slope for different wall functions

5.4 TURBULENCE MODELS

In Chapter 4.3.1 the method of turbulence modelling was described and it was chosen to use realisable k-epsilon as it would result in better modelling of the velocities for rotational water jets. As this choice is subject to different opinions a comparison was done to several other turbulence models.

Two other RANS-models are tested, the default k-epsilon model and the RNG k-epsilon model. Although the basic k-epsilon is the most used model, it is expected to not perform very well for the swirling motion. The RNG k-epsilon model should be able to model this similar to the realisable k-epsilon model, but has disadvantages for round jets [Andersson, et al., 2012]. Application of the k-omega model and the SST-model, which combines k-omega and k-epsilon, were also carried out, but resulted in unstable simulations at the actuator disc. A possible reason for this instability was not found.

Besides these RANS calculations, also several Large Eddy Simulations (LES) were carried out. The use of LES requires a very different model setup as no longer a steady state, but a transient simulation will be done. In a LES simulation the smallest turbulence scales are filtered and only the intermediate-to-large turbulence scales are resolved. The advantage of this method is that the anisotropic large eddies can be directly simulated and are not approximated. However, it has considerably higher computational costs.

In Figure 5-13 the comparison of the velocities is shown for the basic Scenario 6 case and the three comparisons in this piled scenario. This qualitative comparison already shows a difference in the water jets, especially for the low-velocity core.

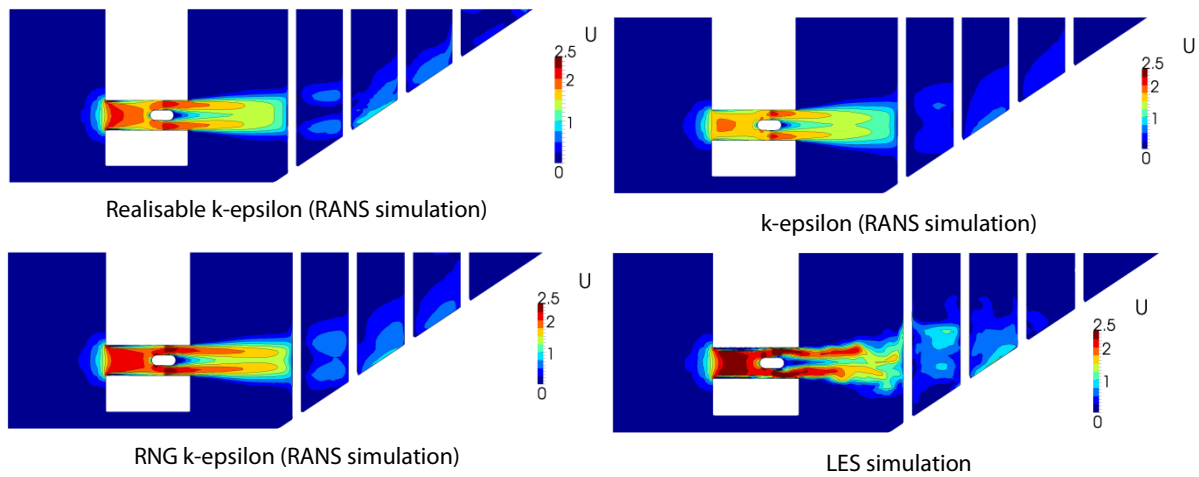


Figure 5-13: Velocity (m/s) plot of different turbulence models at $y/D_0 = 0$. Three steady state RANS simulations are shown as well as an arbitrary time of an LES simulation.

A more quantitative comparison can be seen in Figure 5-14. As a time-step for the LES simulation would only be a snapshot of that moment, the velocities are averaged over a period of time after a start-up time. The deviation of the time-steps for this mean value results in the turbulence kinetic energy with Equation (3.2) and (3.3). The results show a larger low-velocity core for both the RNG k-epsilon simulation, as well as the LES simulation, which was also visible in the measurements in Figure 4-11.

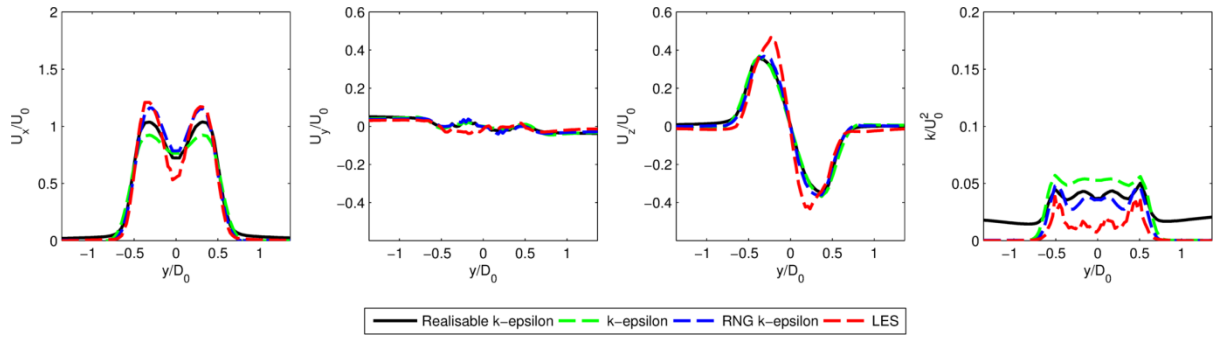


Figure 5-14: Efflux velocities for different turbulence models

As especially the LES model agrees so well, it was also compared to the measurements at the slope in Figure 5-15. These figures show the entire range of measured velocities after the start-up time in the three directions. The shaded area shows the standard deviation (or turbulence) and the dotted line shows the maximum measured value in the written time intervals that could be compared. It shows that the range of results includes nearly all measured average velocities, but the same errors as shown in Chapter 4.4.6 also appear in the LES simulation, although to a smaller extend. Still the velocities in the measurements are higher near the toe.

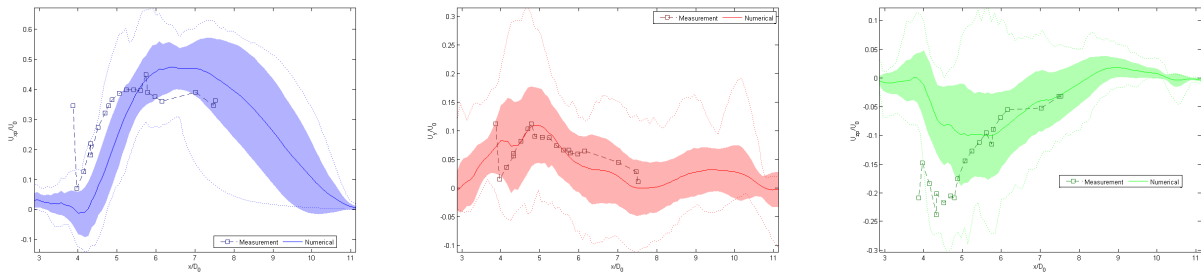


Figure 5-15: Velocities on the slope for the measurement data and the LES numerical results at $y/D_0 = 0.7$ in axial parallel direction (left) lateral direction (centre) and in perpendicular direction (right). The shaded area is the standard deviation, the dotted lines show the maximum and minimum values. The measurement data are shown as squares.

A LES calculation also allows us to look at the change in time in Figure 5-16. This shows that the water jet has its impact at around 1.5 seconds into the simulation. At this moment the highest maximum velocities occur, which reduce when the flow is established. As in reality the bow thruster needs a certain time to advance to full propulsion, the water jet will not impact at full power and this peak will not occur.

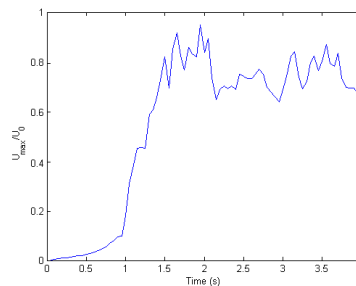


Figure 5-16: Maximum velocity on the slope over time

5.5 EFFLUX VELOCITIES

The axial and tangential coefficients are changed and result in different efflux velocities. For each of these velocities the dimensionless velocities on the slope are computed and shown in Figure 5-17. For the efflux velocities close to the original velocity (1.41, 1.59 and 1.75 m/s) the resulting dimensionless slope velocities are nearly equal, which proves both the scaling of the model for different velocities to give proper results, as well as the independence of the theory of Blaauw and van der Kaa to the efflux velocity.

The largest deviation can be noticed for the lowest efflux velocity where the dimensionless slope velocity has decreased. The diffusion of the water jet in the numerical water jet is in this case higher than the results for the other efflux velocities and is possibly incorrect.

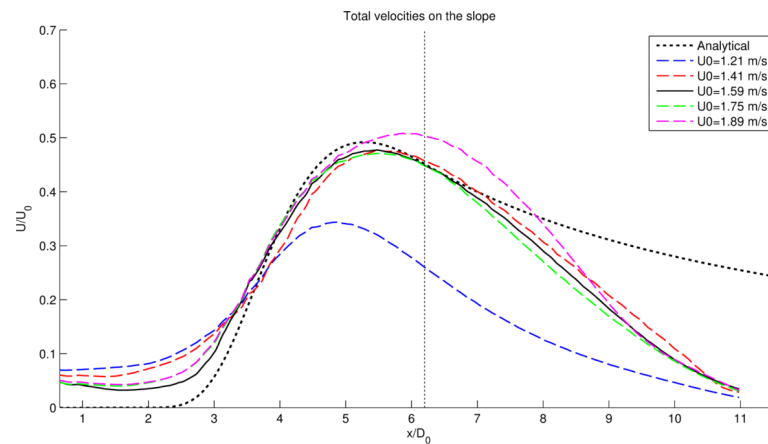


Figure 5-17: The dimensionless total velocities over the slope at $y/D_0 = 0$ for several efflux velocities compared to the theoretical velocities by Blaauw and van der Kaa

5.6 CONCLUSION - SENSITIVITY OF THE MODEL

Sensitivity analysis of the model set-up shows that the generated actuator disc generates a stable efflux. Changes in the mesh, disc thickness or location only show the slight changes as expected. The actuator disc is not influenced by changes of the geometry and performs equal for a sloped scenario and for an unconfined outflow.

Amplifying the actuator disc with a radial component to increase the radial diffusion only results in a larger low-velocity core and a higher dissipation of the total turbulence kinetic energy and does not result in any increase of the radial velocities at the efflux. Variation to the tangential body force coefficient show that the tangential component is necessary for a correct calculation of the velocities on the slope around piles. When no local geometries like piles are present in the close proximity of the thruster, the addition of tangential velocities is less essential.

Different implemented wall functions of the slope and meshes at the slope are compared and show a wide variability of results. As the different meshes in the near wall region resulted in different non-dimensional wall distance, this resulted in different applications of the wall functions. Although it was expected that all results within the prescribed wall distances would

give similar results, none of the simulations converged to one specific solution. Validation of the numerical in a vertical column was not possible as these measurements were only done at one location at the propeller axis on the slope, where all wall functions showed the same results. Comparison to the velocity profile in a simple flume would make it possible to select a correct wall function, but due to time constraints this was not taken into account and the default wall function is used.

Another possible improvement was found by changing the turbulence model. While the realisable k-epsilon model showed the best results of the tested RANS-models, it performed less than the LES model. In a Large Eddy Simulation model more detail of the turbulence is computed instead of approximated and the velocity profile shows higher velocities towards the toe. As LES models require a large running time, they should only be applied in cases where the flow is locally obstructed by structures.

6 RESULTS FOR CHANGING GEOMETRY

The numerical model, as discussed in previous chapters, is used for measuring the velocities in different geometries. At first the velocities at an open quay wall are calculated for different distances to the slope (*Chapter 6.1*) and different slope angles (*Chapter 6.2*) making a comparison possible to the theoretical equations at a slope. Next, the slope is subjected to different alignments of piles to the bow thruster (*Chapter 6.3*). A comparison is made between the velocity increase as concluded by Van Doorn and the velocity increase in the numerical model due to the piles.

Subsequently the velocities at the bottom of a vertical quay wall are determined (*Chapter 6.4*) and the relationship of oblique walls to the downward velocities of the vertical quay wall is compared to the equation of Römisch as shown in Chapter 2.3.5 (*Chapter 6.5*).

At last the numerical scale model is enlarged for a comparison between the scale model and full-size simulations to look if scaling results into any changes in the dimensionless results. (*Chapter 6.6*)

6.1 DISTANCE TO THE SLOPE

In Chapter 2.3.3, Equation (2.23) - (2.25) show that the velocities on the slope are influenced by the distance to the slope. In Figure 6-1 for five distances to the slope the maximum velocities are derived with their location on the slope. It can be seen that for the closer distances the intersection the maximum velocities are measured close to the propeller axis.

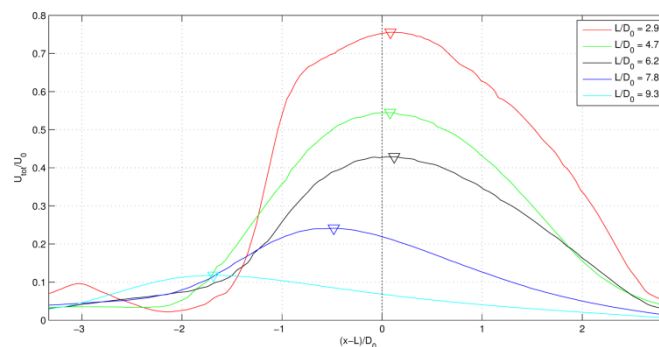


Figure 6-1: Velocities on the slope for different distances to the slope at $y/D_0 = 0$. The triangles indicate the location where the maximum velocities occur.

In Figure 6-2 those maxima are shown in relation to the theoretical velocities of Chapter 2.3.3 and to the scale model measurements of Van Doorn. The shown scenarios of Van Doorn are the scenarios without piles for both the smooth slope (S2 and S3) as well the rough slope (S4

and S5). As only limited measurements are done, these might not include the point of the highest velocity. The factor (f) in the analytical formula is not defined for the 1:1.5 slope with no piles and is not included in the figure [PIANC MarCom, 2013].

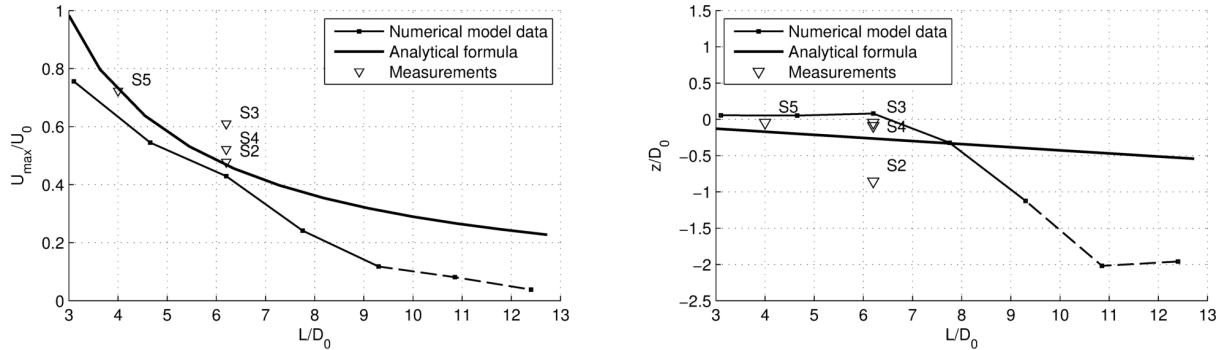


Figure 6-2: (left) Maximum velocities at the slope as a function of the distance to the slope at $y/D_0 = 0$; (right) Location of the highest velocities on the slope of the same runs. The dashed lines indicate the range for which the velocities are at the bottom ($z/D_0 = -2.2$) and the equations no longer applies.

It can be noticed that, although the analytical formula is not amplified by the specified factor, it agrees well with the measurements and only shows a slight underestimate to the measurements in the scenarios. In Scenario 3 and 4 higher velocities were measured. Both had many measurements at the location of impact on the slope and might have measured the highest velocity locations that were not measured in Scenario 2 and 5.

The numerical model underestimates the velocities, as was already expected from Chapter 4.4.6, but show a good estimation and similar trend of decreasing velocities for greater distances. At locations $L/D_0 > 8$, the numerical model shows that the velocities at the bottom have a strong influence on the distribution of the velocities over the slope. For these distances the spreading of the velocities already results in significant velocities over the slope. As a result the location of maximum impact moves to the toe of the structure. For these velocities the theory of Chapter 2.3.3 is no longer valid, as the velocities over the flat water bed are not taken into account in the formula.

6.2 ANGLE OF THE SLOPE

As a second important parameter for the calculation of the velocities on a slope, the angle of the slope is included in Equation (2.26). In Figure 6-3 both the maximum velocities, as well as the location of these maximum velocities, are shown as a function of the angle of the slope at a distance of $L/D_0 = 6.2$. Comparison is made between the measurements by Van Doorn, the numerical model and the theory, without amplification with the factor f.

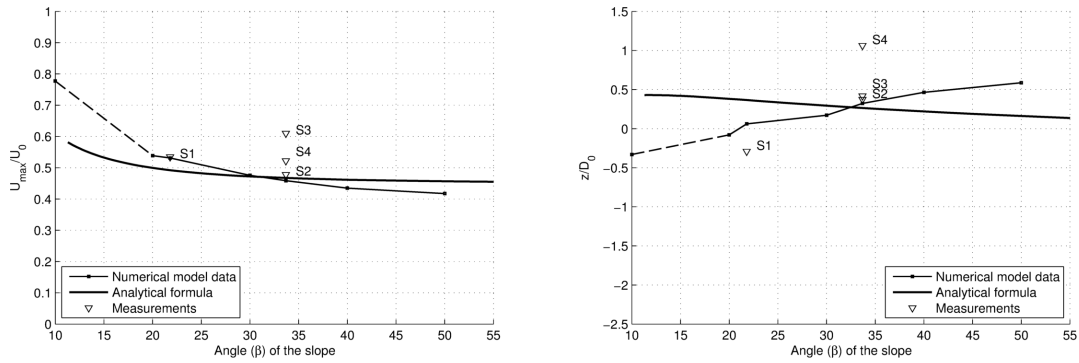


Figure 6-3: Velocities (left) and location of impact (right) for different slope angles. 22 Degrees corresponds with a 1:2.5 slope and 34 degree is equal to a 1:1.5 slope.

The results for the numerical model at an angle of 10 degree is doubtful as the close distance to the slope resulted in a geometry collision between the ship and the slope.

Both the numerical and the physical model show a good agreement with the velocities calculated with the theory of Chapter 2.3.3. Again, the measurements of Van Doorn are slightly higher. The location of highest velocities on the slope is only slightly dependent on the slope and although both models show a higher location for higher angles, while the theory shows a lower location, this is not considered to be significant.

6.3 ADDITION OF PILES

When piles are added to the slope, the flow velocities are expected to increase at a slope with piles as shown in Chapter 2.3.3 and Chapter 2.3.6. Analysis of the RANS model resulted into a questionable simulation and the influence of the alignment of the piles is computed with the use of LES-simulations. In Chapter 5.4 it was concluded this turbulence model will result in better results, especially at structures, but at higher computational costs.

In Figure 6-4 the scenario of *No Piles* is compared to eight different alignments of the piles to the bow thruster. For each scenario the bow thruster is fixed at the location $y/D_0 = 0$, while the piles are moved to their position relative to the pile-diameter ($D_{pile}/D_0 = 0.3$). In all plots the mean velocities are shown for the time span 1.5 - 4 seconds. At the start of this time span (after the starting time), the velocities have already spread for most parts between the piles as shown in Figure 5-16 and although the lateral velocities at the upper part of the slope have not developed yet, the comparison to a longer run showed that this did not influence the results in the piled region.

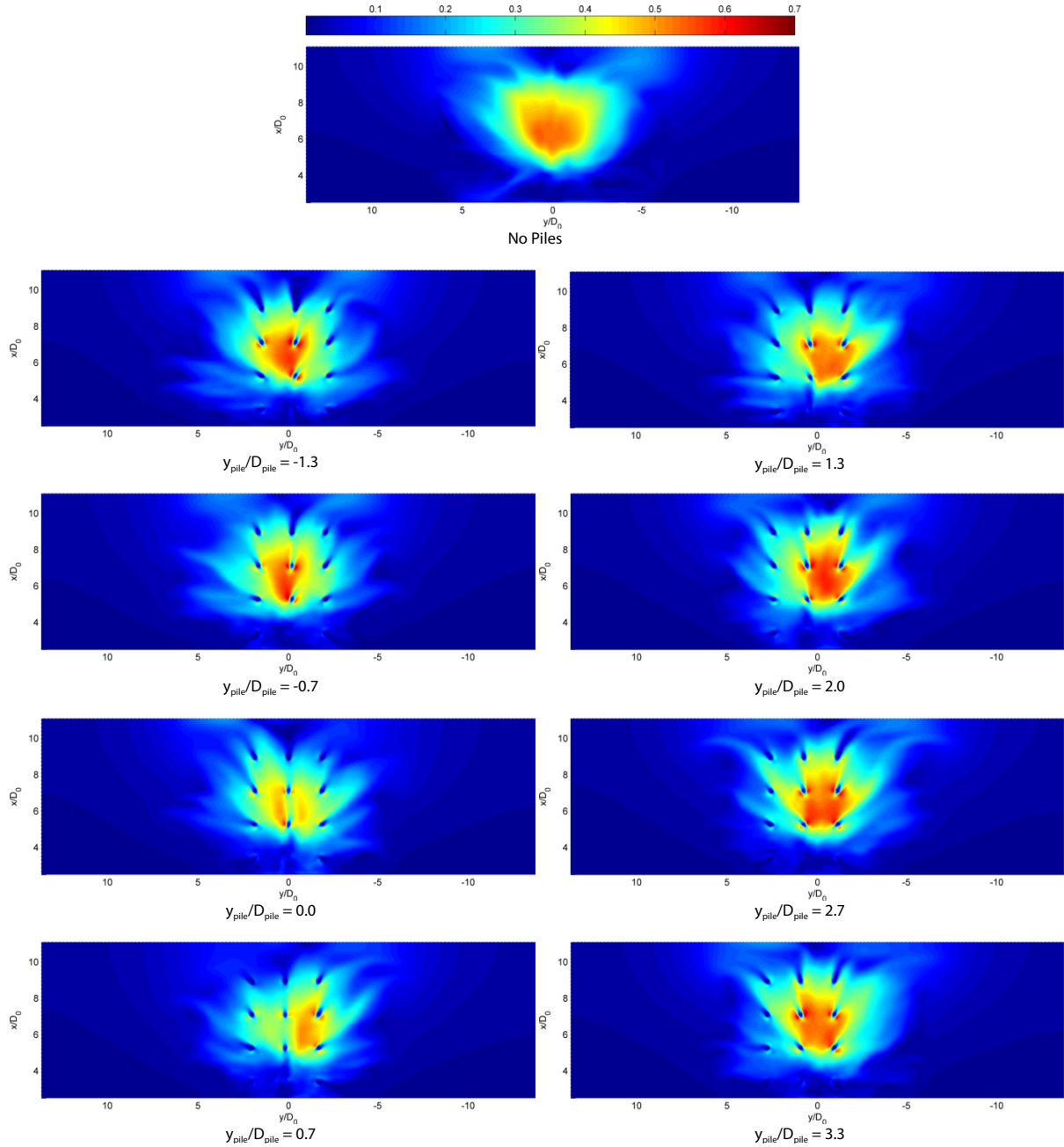


Figure 6-4: Mean velocities U/U_0 at the slope for different distance of the bow thruster to the closest pile. The thruster axis is located at $y/D_0 = 0$ and intersects the slope at $x/D_0 = 6.2$. The location of the piles is shown as points of zero velocity.

Instead of using the mean velocities also the distribution of the velocities can also be used to estimate the maximum velocities. In Figure 6-5 it is shown that, for an arbitrary point with high velocities, the distribution start to approach a normal distribution. For a normal distribution the maximum velocity with an occurrence of 0.1% can be calculated by adding three times the standard deviation. This value is also often used for the calculation of bed protections [Schierck & Verhagen, 2012]. In Appendix I the plots for these velocities are shown as well as a table in which the maximum computed velocities are shown of both figures.

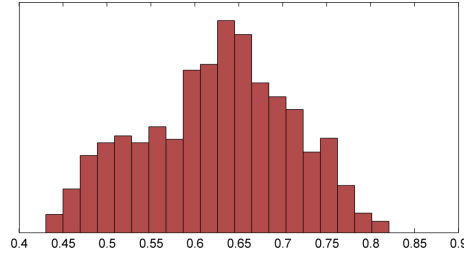


Figure 6-5: Histogram of the measured numerical velocities for an arbitrary point with high velocities

From both Figure 6-4 and Appendix I the conclusion is drawn that the addition of piles on the slope hardly increases the mean and maximum velocities at all. The measured increase at the slope at a distance of D_{pile} from the piles did not exceed 5%. Locally at the piles a far greater increase can be noticed in the order of 35%. However, as the mesh is not adjusted for a LES simulation, it might not be accurate at the piles, since LES simulations require y^+ values in the range of 1 - 10 and will underestimate the turbulent viscosity increase for higher wall distances. This increase is lower than the increase at piles of 100% shown in Equation (2.32), but this equation is meant for a steady approach flow while a water jet has more freedom to flow to other location instead of increasing velocities at the pile.

It can be clearly seen that the lowest velocities at both the slope and the piles are the lowest for $y/D_{\text{pile}} = 0.0$. The effect of the rotation water jet can be noticed when comparing the mirrored situations of $y/D_{\text{pile}} = -0.7$ and $y/D_{\text{pile}} = 0.7$. In the former case the swirling water jet has a downward motion when it hits the pile resulting in higher downward velocities, while in the latter case the water jet is redirection in upward direction. Highest velocities appear when the bow thruster is location at $y/D_{\text{pile}} = 2.0$, where the water jet has its largest increase in the downward velocities.

Comparing maximum velocities for the measurements of Van Doorn as shown in Appendix C.1.4 with the numerical outcome in Figure 6-6 shows a good agreement between both. The drawn conclusions are clearly visible in this plot, but as both the numerical and measured velocities did not accurately measure the velocities near the pile, these might be higher than shown in the figure.

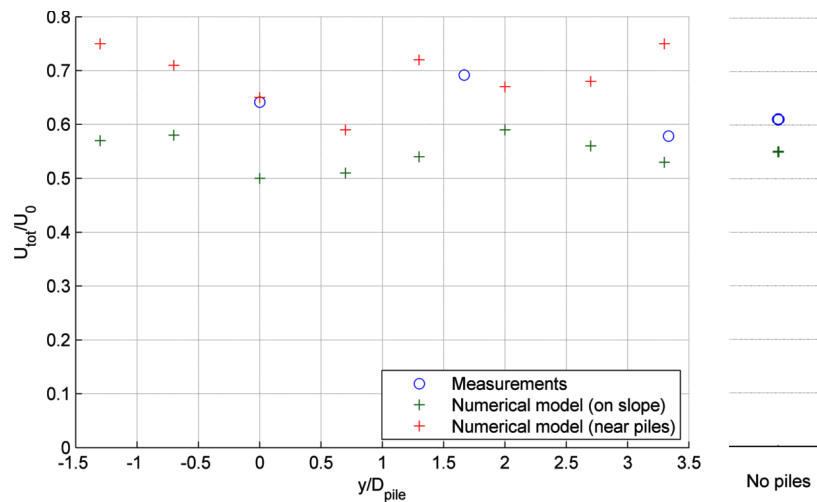


Figure 6-6: Maximum velocities for different pile to bow thruster alignments

6.4 QUAY WALLS

Although the numerical model appeared to show a higher error for steeper slopes, a comparison is done for the velocities at a quay wall and shown in Figure 6-7. Two numerical models have been run with different quay clearances and these are compared to the theoretical maximum bottom velocities as shown in Equation (2.29) and (2.30). This shows that the bottom velocities are largely underestimated in the numerical model, but that the reduction in velocities is similar to the calculated velocities. This underestimate might again be the result of an overestimate in the wall roughness.

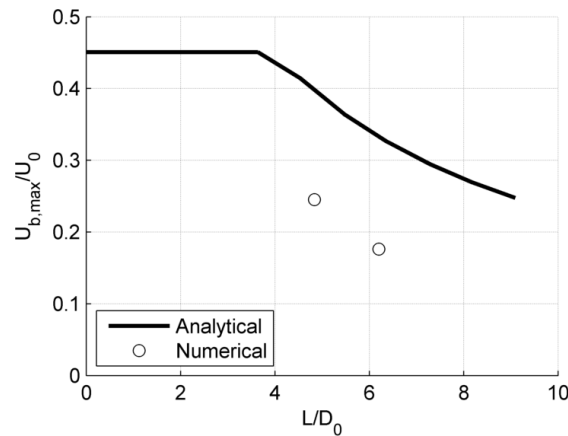


Figure 6-7: Maximum bottom velocities for different distances to the quay wall

Below the ship the limited flow is shown in Figure 6-8. The highest velocities occur for the velocities in axial direction below the duct, but negative axial directions are hardly measured. This in contrast to the measurements done by Van der Laan and Van Blaaderen, which measured higher negative axial velocities [Van der Laan, 2005] [Van Blaaderen, 2006]. These measurements might be influenced by the model setup, as the wall of the basin might have resulted in circulation of the flow, as can be seen in the results of Nielsen in Figure 3-9.

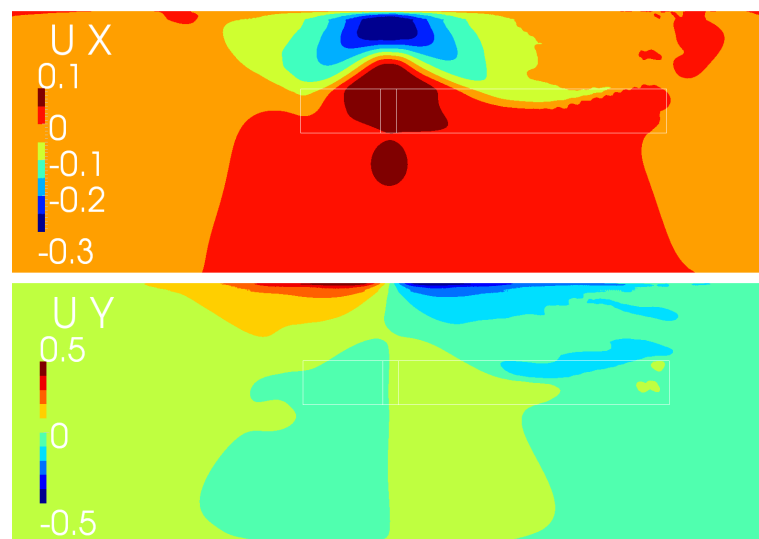


Figure 6-8: Velocities (m/s), $U_0 = 1.59$ m/s. The contour of the 2.5 m long vessel is shown.

6.5 OBLIQUE WALLS

At an oblique wall a reduction factor (C_α) of the downward velocities compared to a vertical quay wall was shown in Chapter 2.3.5. In the numerical model different oblique walls are tested for their maximum downward velocities, resulting in Figure 6-9. This shows a good agreement in the range 0 - 30 degree, but starts deviating at that point. Downward velocities reduce faster than expected and already at an angle of 50 degree the oblique wall acts as a slope with no downward velocities. This is supported by the measurements of Van Doorn's 1:1.5 and 1:2.5 slope. Based on those results it is recommended that the equation of Römisch should not have its maximum validity at 40 degree, but at 30 degree. For larger angles the relations for a slope can be used.

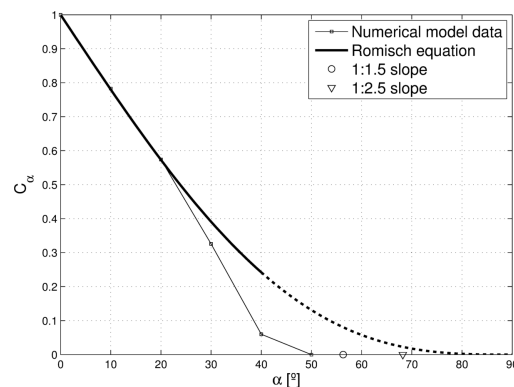


Figure 6-9: Reduction factor for downward velocities compared to a vertical wall ($\alpha=0$). The numerical model data and scale model data for the 1:1.5 and 1:2.5 slope are equal.

6.6 FULL-SIZE MODEL

For use in practise it is required that the numerical also shows proper results for full-size simulations. In Figure 6-10 the comparison of the dimensionless velocities in the scale model of Van Doorn and the full-size model, which is a factor 25 larger than the scale model. For both models the more accurate LES simulation was used. The theoretical solution is added as a reference to earlier figures.

Comparison of both numerical models at the slope, shows that there is a good agreement between both models. The difference can be explained by the short time span used to calculate the mean velocities, which result in an average that is still undergoing changes. It is expected that for longer runs the two models will converge to the same solution.

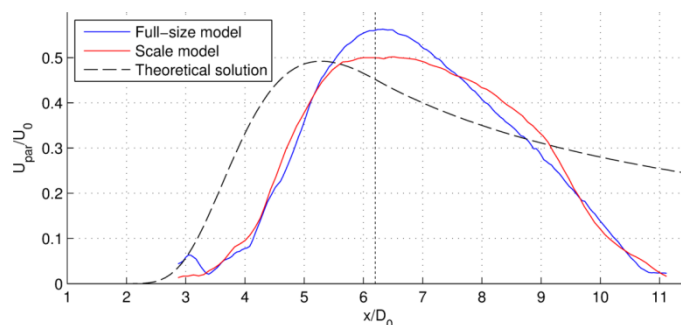


Figure 6-10: Comparison of the velocities in the dimensionless scale model and the full-size model

6.7 CONCLUSION OF THE RESULTS

The comparison of the numerical model to Equation (2.26) for different distances to the slope and for different angles of the slope, shows that the numerical model computes a small underestimate, while the scale model measurements show higher velocities than calculated with the equation. It is shown that in the numerical model a large deviation to the equations occurs for $L/D_0 > 8$, where the highest velocities are measured at the toe and it is concluded that the equation is no longer valid for larger distances as the equation does not take the obstruction by the flat water bed into account.

Different alignments of piles to the bow thruster are compared to a slope without piles for both the mean and the maximum velocities. It is shown that the velocities between the piles on the slopes globally barely increase with the presence of piles (5%). Only the local velocities in the close proximity of piles show an increase up to 35%. This increase is comparable but slightly higher than visible in the scale model measurements, which might be the result of the measured locations not including the exact points with the highest velocities.

At vertical quay walls the numerical model shows a large underestimate of the velocities at the toe of the wall, but a similar trend is visible for increasing quay clearance. The velocities below the ship are only minimal and are not comparable in magnitude with measurements done by Van Blaaderen and Van der Laan. When the vertical quay is slightly angled, the numerical model shows a perfect agreement for the reduction in downward velocities when comparing it to Equation (2.31) of Römisch for oblique quay walls. For angles higher than 30 degree the velocities start to deviate and show lower downward velocities than are calculated with the Equation. It is concluded that the validity of the equation is lower than the prescribed 40 degree and should be reduced to 30 degree.

By magnifying the numerical scale model to a numerical full-size model, the influence of scaling is compared. It is shown that both models show nearly equal dimensionless flow velocities for (short) LES runs. It is expected that for longer runs both models converge to the same solution, which justifies the use of the (scale) model results for the use at open quay structures.

7 CONCLUSIONS AND RECOMMENDATIONS

In this thesis the set-up is made of a three dimensional numerical model to analyse the flow velocity as induced by bow thrusters at a non-erodible bed. In the process of creating this model several conclusions are drawn and more conclusions were drawn by applying and comparing the numerical model to different situations. In this chapter all conclusions are summarised and supplemented with recommendation for further research to improve the set-up or results of the numerical model.

7.1 CONCLUSIONS

Conclusions for the model are separated in the categories *preparations for the numerical model*, *model set-up* and *model results*.

7.1.1 PREPARATIONS FOR THE NUMERICAL MODEL

Scale model research by Van Doorn at the TU Delft resulted in many measurements in different open quay geometries of slopes with and without piles. In the published results an error was created by performing an incorrect rotation of axes. Correction axis rotations are done to the original data and used for the calibration and comparison to the numerical model. (*Chapter 3.1 and Appendix C.1*)

Based on the thrust and torque coefficients and the propeller characteristics an estimation can be made of the velocities at the location of the propeller with the use of the lifting line theory. For this application the propeller geometry is omitted and the vorticity generated at the hub and tip of the propeller blades is not taken into account. The velocities are regarded steady in time with a change over the radius for either a free or ducted propeller. This steadiness was justified in the measurements, as the fine oscillations of the propeller blades could not be measured. (*Chapter 2.1, Chapter 2.2 and Chapter 3.1.3*)

In design practise the formulae for an unconfined yet are corrected for restrictions by a slope with an amplification factor. This factor is based on the incorrectly rotated measurements of Van Doorn and need to be adapted. (*Chapter 2.3.3*)

7.1.2 MODEL SET-UP

The open source software OPENFOAM offers good possibilities for all stages and types of CFD applications. Adaptations to the C++ code are possible, allowing the implementation of a propeller simplification in an actuator disc. The implementation of the propeller in the model does not include the geometry of the propeller itself, as this would result in a computational

expensive velocity field that could not be further calibrated. Instead the propeller velocities are approximated with an actuator disc which locally adds a body force to the momentum equations. A local increase of the turbulence at the hub and the propeller tip is not implemented in the numerical computations. Although a stable second order accurate model is built, the changes to the set-up sometimes result in unexpected or unstable behaviour of the simulation. (*Chapter 4.2, Chapter 4.3 and Chapter 5*)

As the actuator disc is implemented within duct, the usual velocity increase is prevented by the continuity equation. Instead it results in a pressure jump, which subsequently results in a steady flow through the duct as a result of Bernoulli's law. (*Chapter 4.4.1*)

By comparing the implemented Goldstein body force distribution to the measurements the efflux is calibrated. Therefore the axial coefficient is slightly increased to compensate for losses due to wall resistance and the tangential coefficients need a large amplification to correspond with the outflow which is measured by Van Doorn. (*Chapter 4.4.3*)

The boundary layer at the slope depends on the non-dimensional wall distance and the specified wall function for the turbulent viscosity. The sensitivity analysis shows that the velocities are highly depended on the specification of this boundary layer and do not converge to one solution. (*Chapter 5.2.2 and Chapter 5.3*)

7.1.3 MODEL RESULTS

Comparison of the calibrated numerical model shows a very good agreement with the theoretical diffusion of Blaauw and van der Kaa and for a gentle slope with the unamplified measurements theory of Blokland at the slope. These locations also show a good agreement with the scale model measurements. For steeper slopes the numerical simulations deviate from both the theory and the measurements. (*Chapter 4.4.4 - Chapter 4.4.6*)

The simulation shows an underestimate of the maximum velocities when compared to the theoretical and measured maxima. However, when comparing the change of the velocity to changes the distance to the slope and angle of the slope a good comparison can be seen. (*Chapter 6.1 and Chapter 6.2*)

The addition of piles at the slope results in a small increase of the velocities (5%) for locations in between the piles. At locations closer to the piles a higher increase is visible (35%). These maximum velocities at the piles agree very well with the scale model measurements. (*Chapter 6.3*)

The formula of Römisch, describing the downward velocities at an oblique wall, is confirmed with numerical measurements, but a lower maximum angle is advised. At angles larger than 30 degree (slope of 1:0.6) the downward velocities greatly reduce, instead of the 40 degree (slope of 1:0.8) upper bound which is currently prescribed. At slightly higher angles a minimal downward velocity can still be noticed, but the behaviour of the velocities can be better approximated as a slope. (*Chapter 6.5*)

Scaling the model to a full-size simulations shows that the dimensionless velocities agree very well with the numerical scale model. It can be concluded that the model can be used very well for the calculation of velocities at open quay structures, especially for the more gentle slopes. (*Chapter 6.6*)

7.2 RECOMMENDATIONS

The tangential coefficient of the actuator disc needs a very high increase to correspond with the torque in the measurements. The physical and mathematical background of this tangential body force should be further elaborated to determine this necessary increase. (*Chapter 4.4.3*)

For the current simulations the default wall boundary condition is applied. Changes to either the wall function or the mesh near the wall showed a large deviation in resulting velocities. To make a reasoned choice, calibration should be done to a steady velocity profile in a flume. (*Chapter 5.3*)

Most computations were done with a variant of the fast and efficient k-epsilon RANS turbulence model. Although these computations are accurate for simple geometries, it is recommended to use the Large Eddy Simulations for complex flow at structures. (*Chapter 5.4 and Chapter 6.3*)

The generated velocities model can be extended with a calculation of the erosion at the slope. As in those simulations a transient solver is used, the location of the ship no longer has the need to be stationary and can be moved away from the wall over time.

Amplification of the model to a full scale proves the similarities with the scale models. However, for a higher accuracy of the wall boundary, the grid requires a higher level of refinement near the structures and a more computational expensive model. (*Chapter 6.6*)

The model underestimates the velocities at the toe for steeper slopes and vertical quay walls. A clear origin of this inaccuracy is not found and needs to be determined. The radial spreading of the water jet might possibly be influenced and improved with the addition of a free water surface with the Volume of Fluid method.

REFERENCES

- Albertson, M., Dai, Y., Jensen, R. & Rouse, H., 1948. *Diffusion of submerged jets*. New York, American society of civil engineers.
- Andersson, B. et al., 2012. *Computational Fluid Dynamics for Engineers*. Cambridge: Cambridge University Press.
- Blaauw, H. & van de Kaa, E., 1978. *Erosion of bottom and sloping banks caused by the screw race of manoeuvring ships*. Antwerp, Delft Hydraulics laboratory.
- Breusers, H., G.Nicollet & H.W.Shen, 1977. Local scour around cylindrical piers. *Journal Hydraulic Research*, 15[3].
- Buchoux, F., 1995. *Improved algorithms for the computation of induced velocities in propeller design*, s.l.: Massachusetts Institute of Technology.
- Coney, W., 1989. *A method for the design of a class of optimum marine propulsors*. Massachusetts: Massachusetts Institute of Technology.
- De Jong, M., 2003. *Schade veroorzaakt door boegschroeven van manoeuvrerende schepen bij verticale kademuuren*, sl: Delft University of Technology.
- Epps, B., 2010. *OpenProp v2.4 Theory Document*, Massachusetts: Massachusetts Institute of Technology.
- FINE(tm)/Marine, n.d. *Note on the Body Force Propeller implementation*, s.l.: s.n.
- Goldstein, S., 1929. *On the vortex theory of screw propellers*, Cambridge: Proceedings of the Royal Society of London, A, Vol. 123.
- Hough, G. & Ordway, D., 1964. *The generalized actuator disk*, Ithaca: Therm Advanced Research.
- Kärholm, F., 2006. *Rhie-Chow interpolation in OpenFOAM*, Göteborg, Sweden: Chalmers Univesity of Technology.
- Mikkelsen, R., 2003. *Actuator disc methods applied to wind turbines*, Lyngby, Denmark: Technical University of Denmark.
- Murray, G., 2013. *Rotation about an arbitrary axis in 3 dimensions*, s.l.: s.n.
- Nielsen, B., 2005. *Bowthruster-Induced Damage, a physical model study on bowthruster-induced flow*, s.l.: Delft University of Technology.
- OpenFOAM Foundation, 2013. *User Guide*, s.l.: s.n.

- Patankar, S. & Spalding, D., 1972. A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. *International Journal Heat Mass Transfer*, Volume 15, pp. 1787-1806.
- Patankar, S. V. & Spalding, D., 1972. A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. 15[10].
- Paterson, E., Wilson, R. & Stern, F., 2003. *General-Purpose Parallel Unsteady Rans Ship Hydrodynamics Code: CFDSHIP-Iowa*, Iowa City: The University of Iowa.
- Petit, O., 2007. *OpenFOAM Project: Different ways to treat rotation geometries*, Chalmers: s.n.
- PIANC MarCom, 2013. *Guidelines for scour at berthing structures, related to thrusters*, s.l.: s.n.
- Réthoré, P.-E. & Sørensen, N. N., 2008. *Actuator disc model using a modified Rhie-Chow/SIMPLE pressure correction algorithm. Comparison with analytical solutions*. Burssels, Belgium, EWEA.
- Rhie, C. & Chow, W., 1983. Numerical study of the turbulent flow past an airfoil with trailing edge separation. *AIAA Journal*, 21[11], pp. 1525-1532.
- Roulund, A., Mutlu Sumer, B., Fredsøe, J. & Michelsen, J., 2005. Numerical and experimental investigation of flow and scour around a circular pile. *Journal of Fluid Mechanics*, Volume 534, pp. 351-401.
- Schiereck, G. J. & Verhagen, H., 2012. *Introduction to bed, bank and shore protection*. 2nd ed. Delft: VSSD.
- Schokking, L., 2002. *Bowthruster-induced Damage*, s.l.: Delft University of Technology.
- Schottel, n.d. *Transverse thruster, indispensable manoeuvring aid*, Rhein, Germany: s.n.
- Shih, T.-H. et al., 1994. *A new $k-\epsilon$ viscosity model for high Reynolds number turbulent flows - Model development and validation*, Cleveland: National Aeronautics and Space Administration (NASA).
- Shih, T.-H. et al., 1995. A new k -epsilon eddy viscosity model for high Reynolds number turbulent flows: Model development and validation. 24[3].
- Stern, F., Kim, H., Patel, V. & Chen, H., 1988. A Viscous Flow Approach to the Computation of Propeller-Hull Interaction. 32[4], pp. 246-262.
- Stubblefield, J., 2008. *Numerically-based ducted propeller design using vortex lattice lifting line theory*. Massachusetts: Massachusetts Institute of Technology.
- Svenning, E., 2010. *Implementation of an actuator disk in OpenFOAM*, s.l.: Chalmers University of Technology.
- Sweby, P., 1984. High Resolution Schemes Using Flux Limiters for Hyperbolic Conservation Laws. *SIAM Journal on Numerical Analysis*, 21[5], pp. 995-1011.
- Triantafyllou, M. & Franz, S., 2003. *Maneuvering and Control of Marine Vehicles*. Massachusetts: s.n.

- Van Blaaderen, E., 2006. *Modelling bowthruster induced flow near a quay-wall*, s.l.: Delft University of Technology.
- Van der Laan, T., 2005. *Het ontwikkelen van een model voor boegschroefstralen bij verticale kademuren*, s.l.: Delft University of Technology.
- Van Doorn, R., 2012. *Bow Thruster Currents at Open Quay Construction on Piles*, s.l.: Delft University of Technology.
- van Rhee, C., 2010. Sediment Entrainment at High Flow Velocity. *Journal of Hydraulic Engineering*, 136[9].
- Van Veldhoven, V., 2002. *Vooronderzoek schroefstraal op een talud met breuksteen*, sl: Delft University of Technology.
- Verheij, H., 1985. *Aantasting van dwarsprofielen in vaarwegen: Schroefstralen en de stabiliteit van bodem en oevers onder invloed van de stroomsnelheid in de schroefstraal: verslag modelonderzoek*, Delft: Deltares (Waterloopkundig Laboratorium).
- Versteeg, J. & Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics: The finite volume method (Second Edition)*. Harlow: Pearson Education Limited.
- Wolfgang, C., 2011. *Study explaining the fundamental differences between surface and submerged wakes using computational fluid dynamics*, s.l.: The Pennsylvania state university schreyer honors college.

NOMENCLATURE

A	Area	$[m^2]$
A	Coefficient in the Dutch design approach	$[-]$
A_x	Axial coefficient	$[N/m^3]$
A_θ	Tangential coefficient	$[N/m^3]$
C_α	Reduction factor for downward velocities compared to a vertical wall	$[-]$
C_D	Drag coefficient	$[-]$
D	Propeller diameter	$[m]$
D_p	Propeller diameter	$[m]$
D_0	Water jet diameter	$[m]$
F	Vector of the total body force, consisting of f_x and f_θ	$[N/m^3]$
J	Propeller advance ratio	$[-]$
K_T	Thrust coefficient	$[-]$
K_Q	Torque coefficient	$[-]$
L	Distance to the slope	$[m]$
L	Distance of actuator disc to the centre of the duct	$[m]$
P	Engine power	$[W]$
Q	Discharge (water jet)	$[m^3/s]$
Q	Torque	$[Nm]$
R	Propeller radius	$[m]$
R	Radial velocity constant (Chapter 5.1.3)	$[-]$
R_h	Hub radius	$[m]$
Re	Reynolds-number; indicating the ratio between inertial and viscous forces	$[-]$
T	Thrust	$[N]$
U	Velocity (Free stream velocity)	$[m/s]$
U_0	Efflux velocity, velocity in front of the propeller	$[m/s]$
$\langle U \rangle$	Mean velocity	$[m/s]$
V^*	Total velocity (Chapter 2)	$[m/s]$
V	Effective inflow component (Chapter 2)	$[m/s]$
Z	Number of propeller blades	$[-]$
a	Coefficient in the Dutch design approach	$[-]$
b	Coefficient in the Dutch design approach	$[-]$
c_i	Fitting coefficients for the ducted curvature	$[-]$
f	Slope velocity correction factor	$[-]$
f_x	Axial body force	$[N/m^3]$
f_θ	Tangential body force	$[N/m^3]$
h_{pb}	Height between propeller axis and the bed	$[m]$
k	Turbulence kinetic energy	$[m^2/s^2]$

m	Slope, 1:m	[-]
n	Propeller rotations in revolutions per minute	[-/min]
n	Number of velocity measurements	[-]
p	Pressure	[N/m ²]
r	Radial distance	[m]
r	Ratio of the difference with the upstream cell, to the difference with the downstream cell	[-]
r^*	Relative radius	[-]
t	Time	[s]
u	Fluctuating part of the velocity (turbulence)	[m/s]
u^*	Induced velocities (chapter 2)	[m/s]
u_*	Shear velocity	[m/s]
v	Coordinate of the rotational axis for the rotational matrix	[m]
x	Distance	[m]
y	Distance from the efflux axis to the heart of the closest pile	[m]
y^+	Non-dimensional wall distance	[-]
α	Oblique wall angle to a vertical wall = $90 - \beta$	[deg]
β	Slope angle = $\arctan(1/m)$	[deg]
β_i	Angle to plane of rotation (Chapter 2)	[rad]
Δ	Actuator disc thickness	[m]
Γ	Curvature	[m ² /s]
Ω	Propellers angular velocity	[rad/s]
Ψ	Flux limiter function	[-]
ε	Dissipation rate	[m ² /s ³]
μ	Dynamic viscosity	[Ns/m ²]
ρ	Density (of water)	[kg/m ³]
σ	Standard deviation	[-]
ω	Specific turbulence dissipation rate ($=\varepsilon/k$)	[1/s]
ω	Propeller rotation (chapter 2)	[rad/s]
φ	Rotation angle for the rotational matrix	[rad]
ν	$= \mu/\rho$; Kinematic viscosity	[m ² /s]
ν_t	Turbulent kinematic viscosity	[m ² /s]

DIRECTIONAL SUBSCRIPTS

x	Axial direction
y	Lateral direction
z	Vertical direction
x_p	Axial direction parallel to the slope
z_p	Vertical direction perpendicular to the slope
p	Parallel direction on the slope (x_p and y combined)
θ	Tangential direction
r	Radial direction

ABBREVIATIONS

ADV	Acoustic Doppler velocimetry
CFD	Computational Fluid Dynamics
D	Dimensions
EMS	Electromagnetic Suspension
S	Scenario (of Van Doorn)

LIST OF TABLES

Table 3-1: Sensitivity analysis of velocities and turbulent velocities to the <i>Exclude</i> filtering coefficient in [m/s].....	27
Table 4-1: Parameters used in the propeller dictionary.....	34
Table 4-2: Size of cells for the different refinement levels.....	39
Table 4-3: Boundary conditions.....	40
Table 5-1: Wall functions for the turbulent viscosity.....	61
Table A-1: Advantages and disadvantages of several turbulence models, based on [Andersson, et al., 2012].....	87
Table A-2: Parameters for the realisable k-epsilon model.....	88
Table A-3: Scaling from the prototype to the scale model.....	91
Table A-4: Specific parameters of the scenarios of Van Doorn.....	92
Table A-5: Maximum flow velocities in the measurements of Van Doorn after correction for axis rotation error.....	94
Table A-6: Setup of the discretization schemes in fvSchemes.....	105
Table A-7: Simple algorithm and relaxation factors in the fvSolution dictionary.....	106
Table A-8: Linear solvers in the fvSolution dictionary.....	106
Table A-9: Maximum mean velocities in the numerical for different alignments of the bow thruster to the piles.....	114

LIST OF FIGURES

Figure 1-1: (left) Working principle of a bow thruster; (right) De-berthing manoeuvre at a quay structure.....	1
Figure 1-2: (left) Jet spreading at a closed quay wall; (right) Scour holes at a quay wall [PIANC MarCom, 2013]	2
Figure 1-3: Open quay structure supported by piles over a sloped bed [PIANC MarCom, 2013].....	2
Figure 2-1: 2D and 3D view of a transverse (bow) thruster [Schottel, sd].....	5
Figure 2-2: Principle of a pump jet thruster	6
Figure 2-3: (left) A bow thruster in a small cruiser yacht; (right) Four bow thrusters in a very large cruise vessel.....	6
Figure 2-4: Positive axis definitions referred to as axial (x), lateral (y), vertical (z), radial (r) and tangential (θ) direction...	7
Figure 2-5: Cavitation at a propeller	7
Figure 2-6: Ducted propellers with (left) traditional propeller blades and (right) Kaplan type propeller blades	8
Figure 2-7: Decomposition in bound and free vortices of Hough, et al. [Buchoux, 1995]	8
Figure 2-8: Comparison of the representative circulation with two examples of the Goldstein optimum	9
Figure 2-9: The conversion of a propeller to an actuator disc.....	10
Figure 2-10: Hough and Ordway's force distribution in axial directions (left) and tangential directions (right)	10
Figure 2-11: Circulation distribution for a ducted propeller for different gaps between propeller blade and duct.....	11
Figure 2-12: Function fitting to the zero-gap ducted propeller.	12
Figure 2-13: Velocity distribution for a ducted propeller in axial (left) and tangential (right) directions	13
Figure 2-14: Control volume for momentum theory propellers. Based on [Blaauw & van de Kaa, 1978].....	14
Figure 2-15: The zone of flow establishment and zone of established flow by [Albertson, et al., 1948]	16
Figure 2-16: Maximum velocity at the bed as a function of the distance to the quay	18
Figure 2-17: (left) Velocity distribution for an oblique wall; (right) Reduction factor at an oblique wall	18
Figure 2-18: Characteristic features of the flow at a pile [Roulund, et al., 2005].....	19
Figure 3-1: Set-up by Van Veldhoven and Schokking for scour at a slope	21
Figure 3-2: Relative velocities in axial direction. Measured for a pressure jet, free propeller and ducted propeller	22
Figure 3-3: (left) The Vetus bow thruster type 2512B used by Van Doorn; (right) Basin geometry of Scenario 10.....	22
Figure 3-4: Velocity measurements of Van Doorn, showing the standard deviation all three directions	24
Figure 3-5: Location of Van Doorn's measurement points and the velocities at those locations.....	24
Figure 3-6: Measured velocities after filtering with an exclude factor of 4.0	26
Figure 3-7: Measurements of the outflow in vertical direction at a closed quay wall in [m/s]	27
Figure 3-8: (left) Modelling of the flow on a slope; (right) Resulting flow velocities for flow at quay wall	28
Figure 3-9: Comparison of the physical (top) and numerical (bottom) scale model at the bottom of the basin	29
Figure 3-10: Calculated vertical velocity field with a core plate of 0.85D [Van Blaaderen, 2006]	29
Figure 4-1: Use of an Arbitrary Mesh Interface with a propeller in OPENFOAM	33
Figure 4-2: Examples of mesh generated by blockMesh (left), Salome (center) and snappyHexMesh (right)	37
Figure 4-3: Half of the grid for scenario 6 of Van Doorn. The numbers show the level of refinement.....	38
Figure 4-4: Close-ups of the grid at the propeller with the actuator disc shown in red (left) and the at a pile (right).	39
Figure 4-5: Flux limiter functions. (Altered from [Versteeg & Malalasekera, 2007]).....	42
Figure 4-6: Change of pressure, velocity and body force for a one dimensional simulation.....	43
Figure 4-7: Change in velocity distribution (top) and the thrust, torque and discharge (bottom) in the duct.	44
Figure 4-8: Comparison of the analytical to the numerical velocities at the actuator disc	45
Figure 4-9: Velocities in the water jet of the Goldstein optimum at a distance (x) of 95 mm from the efflux.	46
Figure 4-10: Comparison of the relation between the coefficients and the thrust/torque for the Goldstein optimum	46
Figure 4-11: Outflow of the calibrated numerical model.	47

Figure 4-12: Diffusion of the water jet as a function of the dimensionless distance to the efflux at $z/D_0 = 0$	48
Figure 4-13: Total velocities on the slope for the theory of Blaauw and van der Kaa and for numerical simulations	49
Figure 4-14: (left) The locations of plots of Scenario 1; (right) Velocities at the slope over the axial distance	50
Figure 4-15: Velocities at the slope over the lateral lines.....	50
Figure 4-16: The locations of the lateral (green lines) plots of Scenario 2	51
Figure 4-17: Velocities at the slope over the lateral lines.....	51
Figure 4-18: (left) The locations of the plots of Scenario 6; (right) Velocities at the slope over the axial distance	52
Figure 4-19: Velocities at the slope over the lateral lines.....	53
Figure 4-20: Two illustrations showing the water jet of the calibrated RANS model in Scenario 1.	54
Figure 5-1: Efflux velocities for different disc thicknesses	56
Figure 5-2: Efflux velocities for different locations of the propeller	56
Figure 5-3: Efflux velocities for different radial coefficients	57
Figure 5-4: Efflux velocities for different tangential coefficients.....	57
Figure 5-5: Velocities at the slope for different tangential coefficients.....	58
Figure 5-6: Meshing within the duct with the resulting y^+ value, the first mesh used in the calibrated model.....	58
Figure 5-7: Efflux velocities for duct meshes.....	59
Figure 5-8: The non-dimension wall distance (y^+) for a changing number of wall layers in the mesh.	59
Figure 5-9: Influence on the number of layers near the wall, and thereby different wall distances, to the velocities	60
Figure 5-10: Efflux velocities for open outflow and outflow over a slope.....	61
Figure 5-11: Parallel slope velocities at five locations on the slope.....	62
Figure 5-12: Parallel velocities on the slope for different wall functions	62
Figure 5-13: Velocity (m/s) plot of different turbulence models at $y/D_0 = 0$	63
Figure 5-14: Efflux velocities for different turbulence models.....	64
Figure 5-15: Velocities on the slope for the measurement data and the LES numerical results at $y/D_0 = 0.7$	64
Figure 5-16: Maximum velocity on the slope over time.....	64
Figure 5-17: The dimensionless total velocities over the slope at $y/D_0 = 0$ for several efflux velocities.....	65
Figure 6-1: Velocities on the slope for different distances to the slope at $y/D_0 = 0$	67
Figure 6-2: Maximum velocities at the slope as a function of the distance to the slope;	68
Figure 6-3: Velocities (left) and location of impact (right) for different slope angles.	69
Figure 6-4: Mean velocities U/U_0 at the slope for different distance of the bow thruster to the closest pile.	70
Figure 6-5: Histogram of the measured numerical velocities for an arbitrary point with high velocities	71
Figure 6-6: Maximum velocities for different pile to bow thruster alignments.....	71
Figure 6-7: Maximum bottom velocities for different distances to the quay wall	72
Figure 6-8: Velocities (m/s), $U_0 = 1.59$ m/s. The contour of the 2.5 m long vessel is shown.	72
Figure 6-9: Reduction factor for downward velocities compared to a vertical wall ($\alpha=0$).....	73
Figure 6-10: Comparison of the velocities in the dimensionless scale model and the full-size model.....	73
Figure A-1: Axis redefinition in Van Doorn.....	90
Figure A-2: Van Doorn's scenario 6	91
Figure A-3: Dimensions of the scenarios of Van Doorn. x should be read as L. All distances in [mm]	92
Figure A-4: Measurement locations Van Doorn; Axis show the coordinates used by Van Doorn in [mm]	94
Figure A-5: Dimensions of the set-up of Van Blaaderen	96
Figure A-6: Measurement location of Van Blaaderen for the 4 measurement batches.	96
Figure A-7: Implementation of the velocities as a boundary conditions at 1 and 2.....	107
Figure A-8: Dimensionless mean velocities U/U_0 increased with three times the dimensionless turbulent velocities.....	113

APPENDICES

APPENDIX A	TURBULENCE MODELLING	87
A.1	Summary of turbulence models	87
A.2	Equations of the realisable k-epsilon model	88
APPENDIX B	EQUATIONS OF HOUGH AND ORDWAY	89
APPENDIX C	SCALE MODEL RESEARCHES	90
C.1	Open quay structures: Van Doorn	90
C.1.1	Correction to the parallel velocities	90
C.1.2	Model and scenario dimensions	91
C.1.3	Measurement locations	92
C.1.4	Maximum measured velocities	94
C.2	Closed quay wall: Van Blaaderen	96
C.2.1	Model dimensions	96
C.2.2	Measurement locations	96
APPENDIX D	CONSIDERED MODELLING SOFTWARE	97
APPENDIX E	BODY FORCE IMPLEMENTATION IN THE SOLVER	99
APPENDIX F	DISCRETISATION SETTINGS	105
APPENDIX G	IMPLEMENTATION AS BOUNDARY CONDITION	107
APPENDIX H	FULL CALIBRATION FIGURES	108
APPENDIX I	RESULTS AT PILES INCLUDING TURBULENCE	113

Appendix A TURBULENCE MODELLING

A.1 SUMMARY OF TURBULENCE MODELS

The advantages and disadvantages of the different turbulence models are summarised in Table A-1. They are sorted for descending computational costs, increasing number of approximations and decreasing accuracy.

Table A-1: Advantages and disadvantages of several turbulence models, based on [Andersson, et al., 2012]

Turbulence model	Advantages	Disadvantages
Direct numerical modelling (DNS)	- No turbulence model - For low Re numbers - Research only	- Huge costs - Huge amount of data
Large-eddy simulation (LES)	- For complex flows and structures in flows - Gives a lot of information	- High costs - Difficult to identify time convergence - Requires additional treatment at no-slip walls
Reynolds stress models (RSMs)	- Applicable for complex flow: - Swirl, flow separation, plane jets - Includes anisotropy	- Expensive - Inaccurate for some flows, due to introduced closures
Two-equation models	- Both the velocity and length scale are predicted with transport equations - Good results for many engineering applications - Robust, economical and easy to apply	- Eddy-viscosity assumption - Isotropic turbulence - Convection and diffusion of stresses are neglected
• Standard $k-\varepsilon$ model	Most widely used and validated	- Round jets, flows involving significant curvature, swirl, sudden acceleration, separation - Low Re regions
• RNG $k-\varepsilon$ model	Improved for swirling flow and flow separation	- Less stable than standard $k-\varepsilon$ model - Round jets
• Realisable $k-\varepsilon$ model	Improved for swirling flows, flow separation and round jets	Less stable than standard $k-\varepsilon$ model
• $k-\omega$ model	- For low Re regions - No wall functions required - Adverse pressure gradients and separating flow	Fine mesh needed close to the wall
• SST model	- Combines the $k-\varepsilon$ model with the $k-\omega$ in the near-wall region. - Often recommended as replacement of the $k-\varepsilon$ model	Fine mesh needed close to the wall

One-equation models	· Cheap solution for some flows · The approximation of the length scale is too restrictive · Transport of length scale is not accounted for
Zero-equation models	· Cheap solution for some flows · Low transverse flow means no turbulence · No transport of turbulent scales · Cannot be used as general turbulence model

A.2 EQUATIONS OF THE REALISABLE K-EPSILON MODEL

The realisable k-epsilon model is written as [Shih, et al., 1995].

$$\begin{aligned}\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) &= \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k + P_b - \rho \epsilon - Y_M + S_k \\ \frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_j}(\rho \epsilon u_j) &= \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_1 S \epsilon - \rho C_2 \frac{\epsilon^2}{k + \sqrt{\nu \epsilon}} + C_{1\epsilon} \frac{\epsilon}{k} C_{3\epsilon} P_b + S_\epsilon\end{aligned}$$

Where

$$\begin{aligned}C_1 &= \max \left[0.43, \frac{\eta}{\eta + 5} \right] \\ \eta &= S \frac{k}{\epsilon} \\ S &= \sqrt{2 S_{ij} S_{ij}}\end{aligned}$$

In OPENFOAM for incompressible flow the notation is slightly different but the underlying equations are equal. The definitions of the different terms is also given in the equations below.

$$\begin{aligned}\underbrace{\frac{\partial}{\partial t}(k)}_{\text{Advection}} + \underbrace{\frac{\partial}{\partial x_j}(\phi k)}_{\text{Convection}} - \underbrace{\frac{\partial}{\partial x_j} \left[\left(\frac{\nu_t}{\sigma_k} + \nu \right) \frac{\partial k}{\partial x_j} \right]}_{\text{Diffusion}} &= \underbrace{G}_{\text{Production}} - \underbrace{\epsilon}_{\text{Destruction}} \\ \underbrace{\frac{\partial}{\partial t}(\epsilon)}_{\text{Advection}} + \underbrace{\frac{\partial}{\partial x_j}(\phi \epsilon)}_{\text{Convection}} - \underbrace{\frac{\partial}{\partial x_j} \left[\left(\frac{\nu_t}{\sigma_k} + \nu \right) \frac{\partial \epsilon}{\partial x_j} \right]}_{\text{Diffusion}} &= \underbrace{C_1 S \epsilon}_{\text{Production}} - \underbrace{C_2 \frac{\epsilon^2}{k + \sqrt{\nu \epsilon}}}_{\text{Destruction}}\end{aligned}$$

The default parameters are used and shown in Table A-2.

Table A-2: Parameters for the realisable k-epsilon model

C_μ	A_0	C_2	σ_k	σ_ϵ
0.09	4.0	1.9	1.0	1.2

Appendix B EQUATIONS OF HOUGH AND ORDWAY

The full equations as published in Hough and Ordway. For the derivation in Chapter 2 the parts at $r \leq R$, $x = 0$ are used

$$U_x = \begin{cases} \frac{Zx\Omega}{4\pi^2 U r^{\frac{3}{2}}} \int_0^R \frac{\Gamma(r_v)}{\sqrt{r_v}} Q'_{-\frac{1}{2}}(\omega_1) dr_v & \begin{cases} IF r > R, -\infty \leq x \leq \infty \\ OR r \leq R, x < 0 \end{cases} \\ \frac{Z\Omega\Gamma(r)}{2\pi U} + \frac{Zx\Omega}{4\pi^2 U r^{\frac{3}{2}}} \int_0^R \frac{\Gamma(r_v)}{\sqrt{r_v}} Q'_{-\frac{1}{2}}(\omega_1) dr_v & IF r \leq R, x > 0 \\ \frac{Z\Omega\Gamma(r)}{4\pi U} & IF r \leq R, x = 0 \end{cases} \quad (B.1)$$

$$U_\theta = \frac{Z\Omega}{4\pi^2 U \sqrt{r}} \int_0^R \Gamma'(r_v) \sqrt{(r_v)} Q_{\frac{1}{2}}(\omega_1) dr_v \quad (B.2)$$

$$U_r = \begin{cases} 0 & \begin{cases} IF r > R, -\infty \leq x \leq \infty \\ OR r \leq R, x < 0 \end{cases} \\ \frac{Z\Gamma(r)}{2\pi r} & IF r \leq R, x > 0 \\ \frac{Z\Gamma(r)}{4\pi r} & IF r \leq R, x = 0 \end{cases} \quad (B.3)$$

With:

$$Q'_{n-1/2}(\omega) = - \int_{-\frac{\pi}{2}}^{\frac{\pi}{2}} \frac{\cos 2n\alpha}{[2(\omega - 1) + 4 \sin^2 \alpha]^{\frac{3}{2}}} d\alpha \quad (B.4)$$

Appendix C SCALE MODEL RESEARCHES

C.1 OPEN QUAY STRUCTURES: VAN DOORN

C.1.1 CORRECTION TO THE PARALLEL VELOCITIES

The axis definition at the slopes was changed to an x-axis parallel to the slope and a z-axis perpendicular to the slope (x_p and z_p), which required the velocities to be converted to these new directions (Figure A-1). However, incorrect goniometry was used and the values need to be corrected.

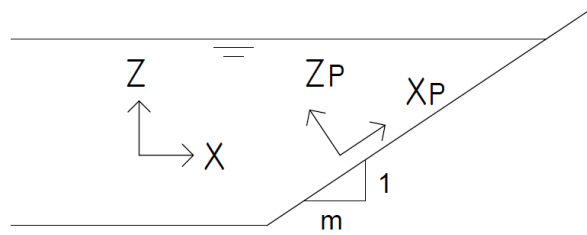


Figure A-1: Axis redefinition in Van Doorn

Van Doorn used:

$$U_{xp} = \frac{1}{\cos\left(\tanh\left(\frac{1}{m}\right)\right)} \cdot U_x \quad (C.1)$$

$$U_{zp} = \frac{1}{\cos\left(\tanh\left(\frac{1}{m}\right)\right)} \cdot U_z \quad (C.2)$$

Where should be used:

$$U_{xp} = \cos\left(\arctan\left(\frac{1}{m}\right)\right) \cdot U_x + \sin\left(\arctan\left(\frac{1}{m}\right)\right) \cdot U_z \quad (C.3)$$

$$U_{zp} = -\sin\left(\arctan\left(\frac{1}{m}\right)\right) \cdot U_x + \cos\left(\arctan\left(\frac{1}{m}\right)\right) \cdot U_z \quad (C.4)$$

This generally reduces the velocities at the slope by 10-30% compared to the data published in his report. The concluding remarks in which it is stated by which factor the hydraulic bed loads are underestimated also change due to these changes. The published values, using an incorrect conversion of the velocities, are still applied for the factor in Chapter 2.3.6.

C.1.2 MODEL AND SCENARIO DIMENSIONS

For the design of the scale model of Van Doorn, two normative vessels were used and several quay constructions worldwide. Based on the smallest possible bow thruster and the dimension of the basin (10 metre by 2 metre), a scaling factor of 25 was chosen. An exceptions was made for the length and width of the vessel, which were reduced to fit in the basin. The vessel was simplified to a squared shape. The slope which is predominantly used can be seen as an upper limit as usually slopes vary in the range of mild slopes of 1:4 to steep slopes of 1:1.2, but this steepness was used to also fit in the basin.

Table A-3: Scaling from the prototype to the scale model

	Prototype	Scale model
Ship length over all	332 [m]	2.50 [m]
Ship width	42.8 [m]	0.30 [m]
Ship draught	14.25 [m]	0.57 [m]
Water depth	15.75 [m]	0.63 [m]
Propeller duct diameter	2.75 [m]	0.11 [m]
Propeller duct length	5.8 [m]	0.30 [m]
Efflux velocity	8.0 [m/s]	1.6 [m/s]
Pile diameter	0.75 [m]	0.030 [m]
Pile distance	5.0 [m]	0.20 [m]

The slope was not constructed over the full width of the basin, but only constructed on a limited width in the centre of the basin. The roughness of the slope and the presence of piles was also only constructed locally at the outflow of the bow thruster. Figure A-2 shows this construction.

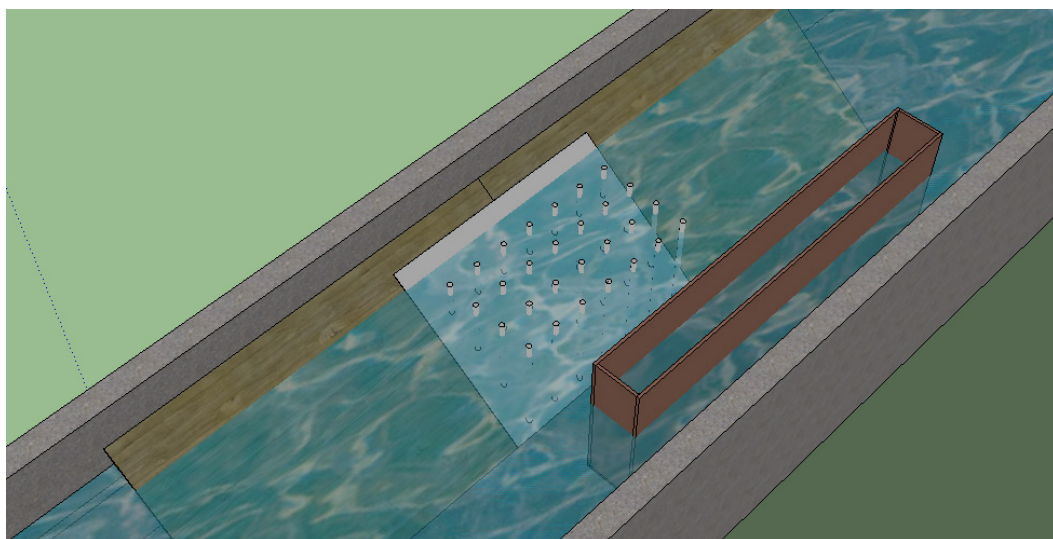


Figure A-2: Van Doorn's scenario 6

In Figure A-3 the dimensions of the scenarios of Van Doorn are shown. A few parameters differed for each scenario: the slope, the distance of the outflow axis to the nearest pile (y) and

the intersection of the outflow axis and the slope (x). These missing parameters are given in Table A-4.

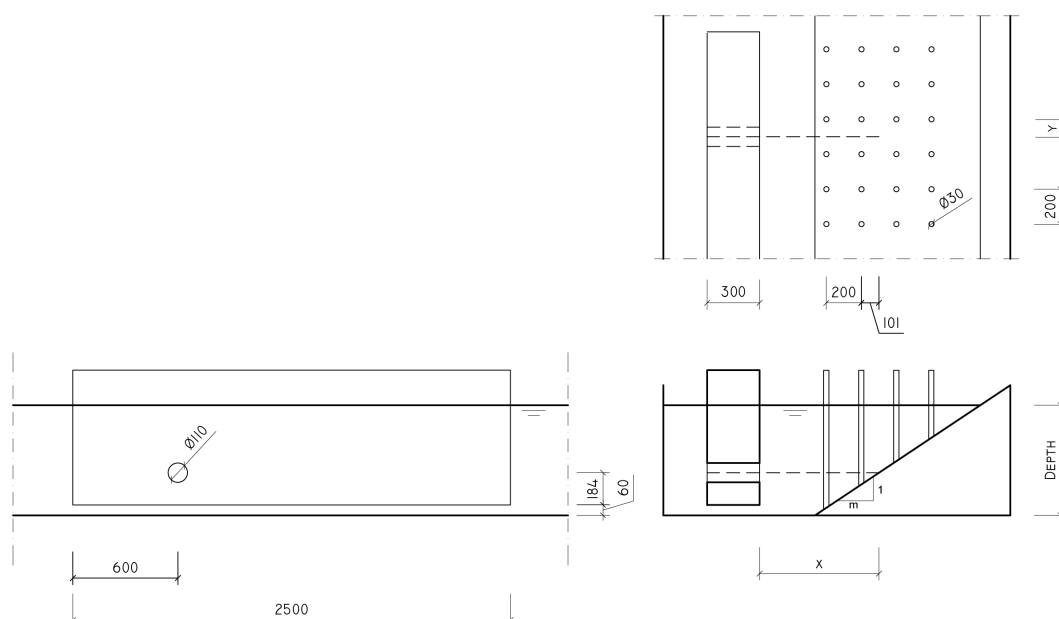


Figure A-3: Dimensions of the scenarios of Van Doorn. x should be read as L. All distances in [mm]

Table A-4: Specific parameters of the scenarios of Van Doorn

Scenario	Slope [1:m]	Depth [mm]	L [mm]	Slope roughness	y [mm]
S1	1:2.5	480	682	Smooth	No Piles
S2	1:1.5	480	682	Smooth	No Piles
S3	1:1.5	630	682	Smooth	No Piles
S4	1:1.5	630	682	Rough	No Piles
S5	1:1.5	630	440	Rough	No Piles
S6	1:1.5	630	682	Smooth	0
S7	1:1.5	630	682	Smooth	100
S8	1:1.5	630	682	Rough	100
S9	1:1.5	630	682	Rough	50
S10	1:1.5	630	682	Rough	0

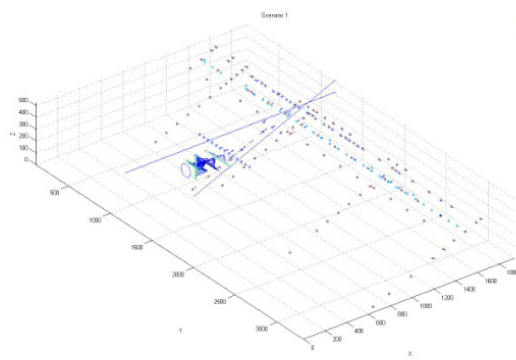
Van Doorn stated in his report that scenario 1 and 2 had a depth of 420 mm, but this conflicted with some of his earlier drawings. As measurements were also at a larger depth, it is expected that the keel clearance was not added in his report and that the actual depth was 480 mm.

C.1.3 MEASUREMENT LOCATIONS

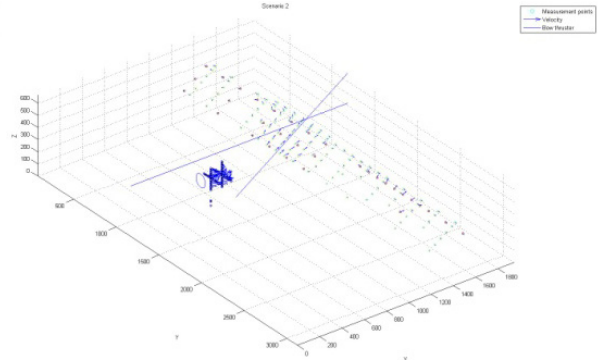
As the available measurement locations are the starting point for the calibrated cases, all locations are given in Figure A-4. In some cases a scenario was constructed several times to do additional measurements. As this resulted in some changes in geometry and the resulting flow

field, the date of the measurements is shown in the colour of the points. Blue being the oldest date and red the newest. The arrows show the flow velocities.

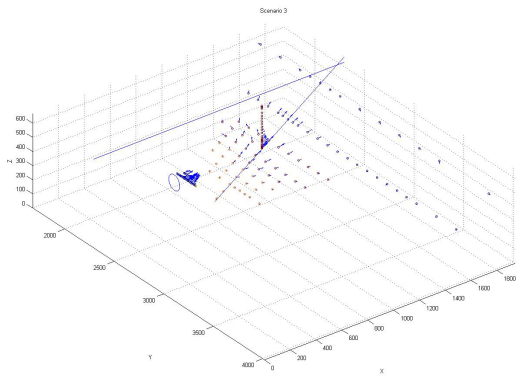
When comparing to the case of Van Doorn one should also take in mind that negative Cartesian axis definitions were used in which the y-axis is oriented different from usual.



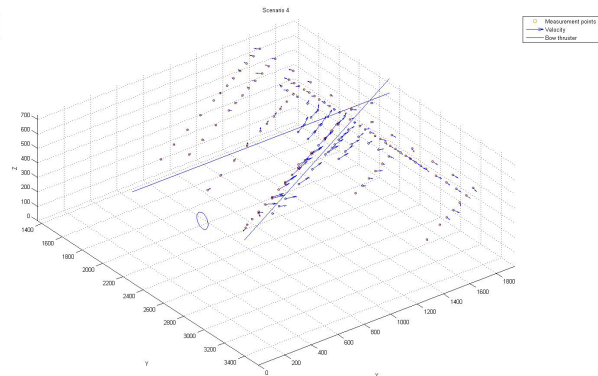
Scenario 1



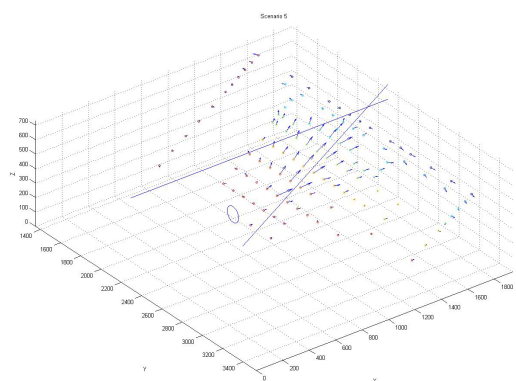
Scenario 2



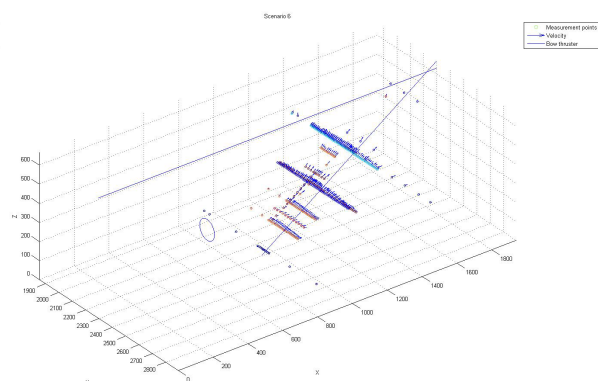
Scenario 3



Scenario 4



Scenario 5



Scenario 6

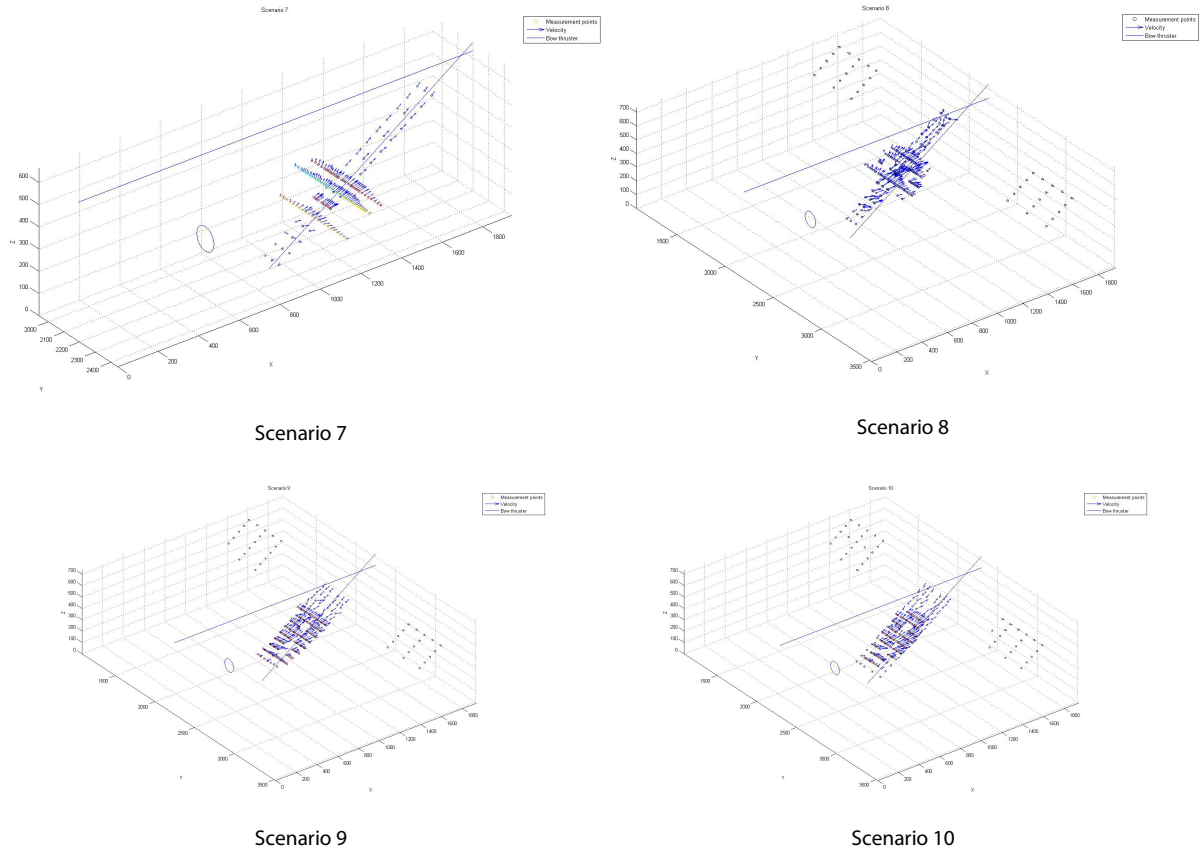


Figure A-4: Measurement locations Van Doorn; Axis show the coordinates used by Van Doorn in [mm]

C.1.4 MAXIMUM MEASURED VELOCITIES

Table A-5: Maximum flow velocities in the measurements of Van Doorn after correction for axis rotation error

	U_{max} [m/s]	k [m^2/s^2]	(x, y, z) to efflux
S1	0.85	0.38	(0.54, 0.04, -0.032)
	0.80	0.22	(0.74, 0.04, 0.048)
	0.63	0.25	(1.04, -0.76, 0.168)
S2	0.76	0.25	(0.726, -0.094, 0.041)
	0.75	0.26	(0.676, -0.094, 0.008)
	0.75	0.21	(0.776, 0.006, 0.075)
S3	0.97	0.33	(0.676, -0.005, 0.046)
	0.86	0.30	(0.676, -0.005, 0.011)
	0.83	1.44	(0.726, -0.005, 0.041)
S4	0.83	0.32	(0.722, -0.005, 0.117)
	0.81	0.29	(0.622, -0.005, 0.05)
	0.81	0.55	(0.522, -0.005, -0.027)
S5	1.15	0.67	(0.33, -0.005, 0.017)
	1.06	0.65	(0.43, -0.005, 0.083)
	1.05	0.38	(0.53, -0.005, 0.15)
S6	0.90	0.27	(0.631, 0.045, -0.004)
	0.90	0.44	(0.631, -0.045, -0.004)
	0.88	0.28	(0.631, 0.055, -0.004)
S7	0.87	0.25	(0.576, 0.045, -0.041)
	0.85	0.26	(0.631, -0.085, -0.004)
	0.77	0.26	(0.676, -0.105, 0.026)
S8	0.92	0.71	(0.672, -0.005, 0.089)
	0.77	0.48	(0.672, -0.005, 0.093)

	0.60	0.71	(0.672, -0.005, 0.083)
S9	1.10	0.69	(0.472, 0, -0.024)
	0.96	0.27	(0.623, 0.005, 0.077)
	0.92	0.50	(0.522, 0, 0.009)
S10	1.02	0.40	(0.622, -0.1, 0.076)
	0.95	1.09	(0.472, 0.05, -0.024)
	0.88	0.34	(0.622, -0.05, 0.076)

C.2 CLOSED QUAY WALL: VAN BLAADEREN

C.2.1 MODEL DIMENSIONS

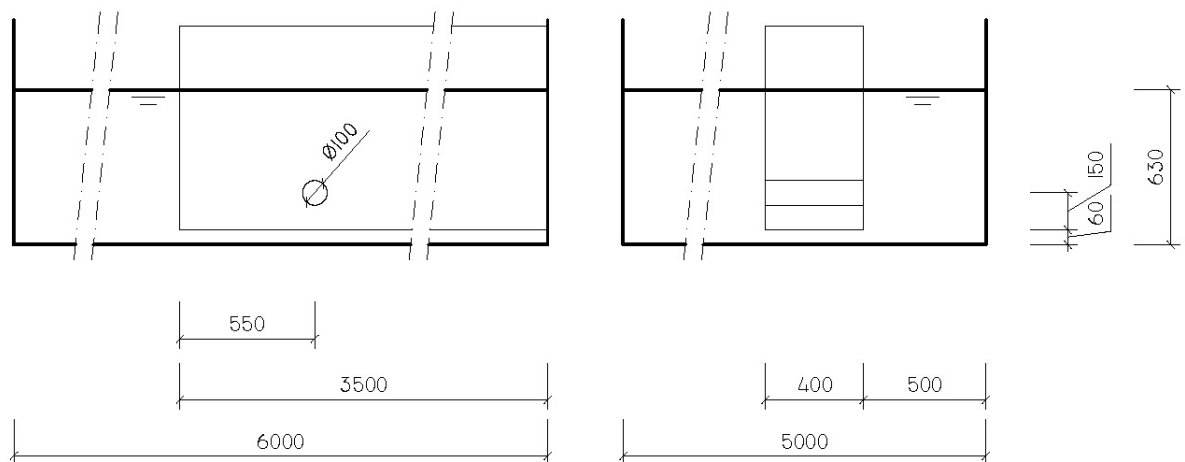


Figure A-5: Dimensions of the set-up of Van Blaaderen

Although the diameter of the bow thruster is approximately equal, the velocities are less than half the velocity generated by Van Doorn.

C.2.2 MEASUREMENT LOCATIONS

Van Blaaderen did his measurements in 4 batches. The locations of each of those is shown in Figure A-6.

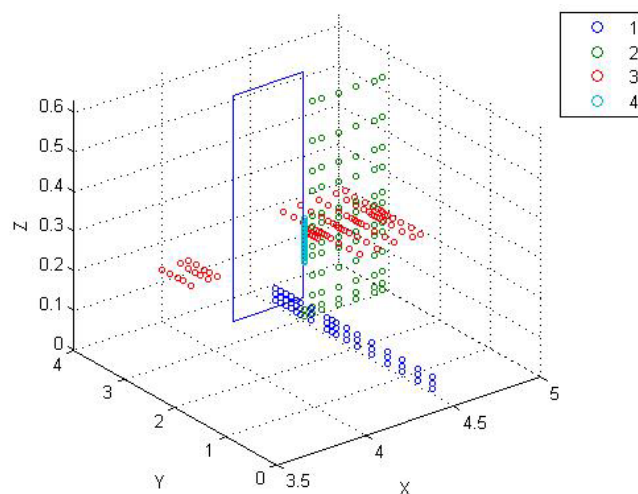


Figure A-6: Measurement location of Van Blaaderen for the 4 measurement batches. The square indicates the modelled vessel at the location of the bow thruster

Appendix D CONSIDERED MODELLING SOFTWARE

For the modelling of a bow thrusters a wide variations of tools is available. Ranging from software aimed at the translations of waves by morphology or generating forces to detailed Computational Fluid Dynamics with a wide application in areas of science and engineering.

Below a short overview is given of the considered packages. This shortlist is limited to software of which sufficient knowledge was available at the office. It consists of three CFD models and two models that have a clearer focus on waves.

PHOENICS

In all previous studies at the TU Delft, PHOENICS was used for the numerical modelling of the bow thrusters induced flows. It is a commercial code developed by CHAM and is fully self contained as it also includes grid generations and post processing in a 3D interface. It is also possible to learn the PHOENICS Input Language to provide input via a text editor. PHOENICS has a user-friendly interface and allows for simple introducing of sink terms and arbitrary equations.

OPENFOAM

OPENFOAM stands for Open Field Operation And Manipulation. It is a free, open source CFD software package produced by OpenCFD Ltd and is being used in most areas of science and engineering. OPENFOAM includes tools for meshing and pre- and post-processing. Advantages of OPENFOAM are the unstructured grid capabilities, the low computational costs due to parallelization and the convenient system for partial differential equations. The used discretization principle is the finite volume method which is accurate for the convection terms.

OPENFOAM has many users worldwide and has solvers for many different problems as well as many available tutorials by different courses given. It however does not have an integrated graphical user interface or a detailed programmer's guide, making the learning curve very gradual. Application of a free water surface is not standard included, but previous research showed that it can be incorporated if necessary.

FINLAB

FinLab is a CFD model developed by ir. R.J. Labeur at Svašek and Delft University of Technology. It solves the Navier-Stokes equations in two and three dimensions for incompressible fluids. FinLab uses the finite element method with fully unstructured grids and also includes a moving free surface. By using the finite elements method the convection terms can be mod-

elled accurate and it has been used for a wide range of hydraulic engineering problems involving complex, small-scale geometries.

As FinLab does not yet include a user friendly interface, it is not available on the web. It is, however, open source software.

SWASH

SWASH (Simulating WAVes till SHore) is an open source numerical tool for simulating free-surface, rotational flows and is generally meant for modelling the complex changes in waves in coastal waters and ports with the use of non linear shallow water equations.

PhD student ir. Rijndorp (TU Delft) stated in a conversation that calculations with a high resolution are possible, but as these are not the aim of Swash measures should be taken to correctly include the turbulence model and the moving propeller. Use of Swash for the modelling of a bow thruster is advised against.

DELFT3D

Delft3D is developed by Deltares and distributed as Open Source software. It is a flexible integrated modelling suite and allows for the creation of complicated three-dimensional flows and the interaction to sediment transport and water quality. It includes morphology, sediment transport and water quality.

There is a wide range of applications for Delft3D and for the necessary fine 3D turbulence the DFlow package might be applicable in the future. However, the current development status does not allow this model to be used for the bow thruster modelling.

Appendix E BODY FORCE IMPLEMENTATION IN THE SOLVER

The scripts have been made as an addition to the normal simpleFoam script. Changes are highlighted. The version shown below is for the OPENFOAM 2.2.1 version.

SIMPLEFOAM.C

```
/*-----*\
=====
\\ / F ield           | OpenFOAM: The Open Source CFD Toolbox
\\ / O peration       |
\\ / A nd              | Copyright (C) 2011-2013 OpenFOAM Foundation
\\ / M anipulation    |
-----*/

License
This file is part of OpenFOAM.

OpenFOAM is free software: you can redistribute it and/or modify it
under the terms of the GNU General Public License as published by
the Free Software Foundation, either version 3 of the License, or
(at your option) any later version.

OpenFOAM is distributed in the hope that it will be useful, but WITHOUT
ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or
FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License
for more details.

You should have received a copy of the GNU General Public License
along with OpenFOAM. If not, see <http://www.gnu.org/licenses/>.

Application
    simpleFoam

Description
    Steady-state solver for incompressible, turbulent flow

/*-----*/

#include "fvCFD.H"
#include "singlePhaseTransportModel.H"
#include "RASModel.H"
#include "simpleControl.H"
#include "fvIOoptionList.H"

// *****

int main(int argc, char *argv[])
{
    #include "setRootCase.H"
    #include "createTime.H"
    #include "createMesh.H"
    #include "createFields.H"
    #include "createFvOptions.H"
    #include "initContinuityErrs.H"
    #include "createPropForce.H"

    simpleControl simple(mesh);

    // *****
}
```

```

Info<< "\nStarting time loop\n" << endl;

while (simple.loop())
{
    Info<< "Time = " << runTime.timeName() << nl << endl;

    // --- Pressure-velocity SIMPLE corrector
    {
        #include "UEqn.H"
        #include "pEqn.H"
    }

    turbulence->correct();

    runTime.write();

    Info<< "ExecutionTime = " << runTime.elapsedCpuTime() << " s"
        << "   ClockTime = " << runTime.elapsedClockTime() << " s"
        << nl << endl;
}

Info<< "End\n" << endl;

return 0;
}

// *****

```

CREATEFIELDS.H

```

Info<< "Reading field p\n" << endl;
volScalarField p
(
    IOobject
    (
        "p",
        runTime.timeName(),
        mesh,
        IOobject::MUST_READ,
        IOobject::AUTO_WRITE
    ),
    mesh
);

Info<< "Reading field U\n" << endl;
volVectorField U
(
    IOobject
    (
        "U",
        runTime.timeName(),
        mesh,
        IOobject::MUST_READ,
        IOobject::AUTO_WRITE
    ),
    mesh
);

#include "createPhi.H"

label pRefCell = 0;
scalar pRefValue = 0.0;
setRefCell(p, mesh.solutionDict().subDict("SIMPLE"), pRefCell, pRefValue);

singlePhaseTransportModel laminarTransport(U, phi);

autoPtr<incompressible::RASModel> turbulence
(
    incompressible::RASModel::New(U, phi, laminarTransport)
);

```

UEQN.H

```
// Momentum predictor
tmp<fvVectorMatrix> UEqn
(
    fvm::div(phi, U)
    + turbulence->divDevReff(U)
    - propForce
    ==
    fvOptions(U)
);

UEqn().relax();

fvOptions.constrain(UEqn());

solve(UEqn() == -fvc::grad(p));

fvOptions.correct(U);
```

PEQN.H

```
{
    volScalarField rAU(1.0/UEqn().A());
    volVectorField HbyA("HbyA", U);
    HbyA = rAU*UEqn().H();
    UEqn.clear();

    surfaceScalarField phiHbyA("phiHbyA", fvc::interpolate(HbyA) & mesh.Sf());
    adjustPhi(phiHbyA, U, p);

    fvOptions.relativeFlux(phiHbyA);

    // Non-orthogonal pressure corrector loop
    while (simple.correctNonOrthogonal())
    {
        fvScalarMatrix pEqn
        (
            fvm::laplacian(rAU, p) == fvc::div(phiHbyA)
        );

        pEqn.setReference(pRefCell, pRefValue);

        pEqn.solve();

        if (simple.finalNonOrthogonalIter())
        {
            phi = phiHbyA - pEqn.flux();
        }
    }

    #include "continuityErrs.H"

    // Explicitly relax pressure for momentum corrector
    p.relax();

    // Momentum corrector
    U = HbyA - rAU*fvc::grad(p);
    U.correctBoundaryConditions();
    fvOptions.correct(U);
}
```

CREATEPROPFORCE.H

```
Info<< "Creating Propeller forcefield\n" << endl;

// Create vector
volVectorField propForce
(
    IOobject
    (
        "propForce",
```

```

        runTime.timeName(),
        mesh,
        IOobject::NO_READ,
        IOobject::AUTO_WRITE
    ),
    mesh,
    dimensionedVector("zero",
                      dimForce/dimVolume/dimDensity,
                      vector::zero)
);

// Read dictionary
IOdictionary propellerDict
(
    IOobject
    (
        "propellerDict",
        runTime.time().constant(),
        runTime,
        IOobject::MUST_READ,
        IOobject::NO_WRITE
    )
);

// Include dictionary constants
const vector propOrigin (propellerDict.lookup("propOrigin"));
vector outflowDirection (propellerDict.lookup("outflowDirection"));

const scalar Rc (readScalar(propellerDict.lookup("coreRadius")));
const scalar R (readScalar(propellerDict.lookup("radius")));

const scalar Ax (readScalar(propellerDict.lookup("Ax")));
const scalar Atheta (readScalar(propellerDict.lookup("Atheta")));
const scalar rhewater (readScalar(propellerDict.lookup("rho")));

const scalar thickness (readScalar(propellerDict.lookup("thickness")));

//Calculate rotation to the (1 0 0) direction
outflowDirection /= (::mag(outflowDirection));

scalar rot = ::acos(outflowDirection[0]);

vector axial = vector(1,0,0);

vector matrixRotVector = axial^outflowDirection;

if (!(outflowDirection==axial)){
    matrixRotVector /= (::mag(matrixRotVector));
}

// Define cells
const scalar pi (M_PI);
scalar numCells(0);
scalar numInside(0);
scalar analyticalVolume (pi*(R*R-Rc*Rc)*thickness);
scalar numericalVolume (0.0);

scalar rcpart (Rc/R);
scalar rs;
scalar rpart;
scalar fx;
scalar ftheta;

scalar axialForce (0.0);
scalar tangentialForce (0.0);

// The rotation matrix (V) shown below is not used as calculating with vectors is a lot more easy
// Instead of a transposed matrix the -inv- vectors are used.
// const tensor V(sqr(matrixRotVector[0])+(1+sqr(matrixRotVector[0]))*::cos(rot),

```

```

// matrixRotVector[0]*matrixRotVector[1]*(1-::cos(rot))-matrixRotVector[2]*::sin(rot),
// matrixRotVector[0]*matrixRotVector[2]*(1-::cos(rot))+matrixRotVector[1]*::sin(rot),
// matrixRotVector[0]*matrixRotVector[1]*(1-::cos(rot))+matrixRotVector[2]*::sin(rot),
// sqr(matrixRotVector[1])+(1-sqr(matrixRotVector[1]))*::cos(rot),
// matrixRotVector[1]*matrixRotVector[2]*(1-::cos(rot))-matrixRotVector[0]*::sin(rot),
// matrixRotVector[0]*matrixRotVector[2]*(1-::cos(rot))-matrixRotVector[1]*::sin(rot),
// matrixRotVector[1]*matrixRotVector[2]*(1-::cos(rot))+matrixRotVector[0]*::sin(rot),
// sqr(matrixRotVector[2])+(1-sqr(matrixRotVector[2]))*::cos(rot));

const vector Vx(sqr(matrixRotVector[0])+(1+sqr(matrixRotVector[0]))*::cos(rot),
matrixRotVector[0]*matrixRotVector[1]*(1-::cos(rot))-matrixRotVector[2]*::sin(rot),
matrixRotVector[0]*matrixRotVector[2]*(1-::cos(rot))+matrixRotVector[1]*::sin(rot));
const vector Vy(matrixRotVector[0]*matrixRotVector[1]*(1-::cos(rot))+matrixRotVector[2]*::sin(rot),
sqr(matrixRotVector[1])+(1-sqr(matrixRotVector[1]))*::cos(rot),
matrixRotVector[1]*matrixRotVector[2]*(1-::cos(rot))-matrixRotVector[0]*::sin(rot));
const vector Vz(matrixRotVector[0]*matrixRotVector[2]*(1-::cos(rot))-matrixRotVector[1]*::sin(rot),
matrixRotVector[1]*matrixRotVector[2]*(1-::cos(rot))+matrixRotVector[0]*::sin(rot),
sqr(matrixRotVector[2])+(1-sqr(matrixRotVector[2]))*::cos(rot));
const vector Vinx = vector(Vx[0],Vy[0],Vz[0]);
const vector Vinvy = vector(Vx[1],Vy[1],Vz[1]);
const vector Vinvz = vector(Vx[2],Vy[2],Vz[2]);

// All cells are processed in a loop.
forAll(mesh.cells(),cellI)
{
    ++numCells;

    const vector iCenter = mesh.C()[cellI];
    vector iDist = (iCenter-propOrigin);

    // Converting of the cells to new coordinate system in which the propeller is located at (1 0
0)
    iDist = vector(Vinx & iDist,Vinvy & iDist,Vinvz & iDist);

    // Calculating radius
    const scalar iR (:: sqrt(sqr(iDist[1])+sqr(iDist[2])));

    // Apply force to the cells within the propeller location
    if( (pos(R-iR)) &&
        (pos(iR-Rc)) &&
        (pos(iDist[0]+thickness/2)) &&
        (pos(thickness/2-iDist[0]))
        ) {
        ++numInside;
        numericalVolume += mesh.V()[cellI];

        // Force formulae
        rpart = iR/R;
        rs = (rpart - rcpart)/(1-rcpart);
        fx = Ax * rs * ::sqrt(1-rs);
        ftheta = Atheta * rs * ::sqrt(1-rs)/(rs*(1-rcpart)+rcpart);

        // Divided by rhowater for incompressible solvers and split into y and z direction
        scalar Faxial = fx/rhowater ;
        scalar Ftany = ftheta/rhowater *-iDist[2]/iR;
        scalar Ftanz = ftheta/rhowater * iDist[1]/iR;
        vector F(Faxial,Ftany,Ftanz);

        // Transforming the body force to the real location
        propForce[cellI] = vector(Vx & F,Vy & F,Vz & F);

        // Calculating the sums to show in the output
        axialForce += fx * mesh.V()[cellI];
        tangentialForce += ftheta * mesh.V()[cellI]*iR;
    }
}

// General information output
Info << "Inside: " << numInside << " of " << numCells << endl;
Info << "Analytical volume: " << analyticalVolume << endl;

```

```
Info << "Numerical volume: " << numericalVolume << endl;
Info << "Axial force: " << axialForce << endl;
Info << "Tangential force: " << tangentialForce << endl;
```

PROPELLERDICT

This file is needed in the case directory. An explanation of the parameters is given in Table 4-1.

```
/*-----*- C++ -*-----*/
|=====|
| \ \ / / F ield | OpenFOAM: The Open Source CFD Toolbox
| \ \ / / O peration | Version: 2.2.0
| \ \ / / A nd | Web: www.OpenFOAM.org
| \ \ / / M anipulation |
|-----|

FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       propellerDict;
}

// Dictionary for the simpleFoamProp

propOrigin      (0.05 0 0);
outflowDirection (1 0 0);
coreRadius      0.018;
radius          0.049;

Ax              2.8e6;
Atheta         -6e+05;
rho            1e3;

thickness       0.005;
```

Appendix F DISCRETISATION SETTINGS

In the fvSchemes file the numerical schemes for terms, such as derivatives in equations, are set. OPENFOAM offers an unrestricted choice to the user. Default values can be applied per category, but it is also possible to assign an entry to each derivative individually. Table A-6 shows the used entries.

As the solver that is used is a steady state solver, the time derivatives need steadyState as entry. Most other schemes are based on the Gaussian finite volume integration method, but since this method needs the values on the cell faces to be known, an interpolation scheme has to be specified.

Table A-6: Setup of the discretization schemes in fvSchemes

Sub-dictionary	Keyword	Entry
ddtSchemes	ddt(epsilon)	steadyState
	ddt(k)	
gradSchemes	grad(U)	Gauss linear
	grad(p)	
	grad(epsilon)	
	grad(k)	
	grad(muEff*dev(T(grad(U))))	
div Schemes	div(phi,U)	Gauss linear
	div(phi,epsilon)	bounded Gauss limitedLinear 1
	div(phi,k)	
	div(phi,muEff)	
laplacianSchemes	laplacian(muEff,U)	Gauss linear corrected
	laplacian((1 U(A)),p)	
	laplacian(DepsilonEff,epsilon)	
	laplacian(DkEff,k)	
interPolationSchemes	interpolate(HbyA)	linear
snGradSchemes	-	
fluxRequired		p

The equation solvers, tolerances and algorithms are controlled from the fvSolution dictionary. In this file the details of the SIMPLE algorithm and the linear solvers are included. In Table A-7 the former is shown. Non orthogonal correctors can be included if the mesh shows a high degree of non-orthogonality. For a mesh with an maximum non-orthogonality to approximately 70 degree there is no need to include any correctors. The residual control influences the length of the simulation. When the residual for every field falls below the corresponding residual, the simulation terminates. The pRef values are necessary to the calculations to the relative pressure difference in this incompressible solver. For a higher stability of the simula-

tion, relaxation factors can be applied which reduce the extent to which the cell value changes between time steps. A value of 1.0 corresponds to the fastest converging method, but often has a high instability. Reducing the factor increases this stability, but also increases the computational costs. The used factors are obtained by trial and error.

Table A-7: Simple algorithm and relaxation factors in the fvSolution dictionary

Sub-dictionary	Keyword	Entry
SIMPLE	nNonOrthogonalCorrectors	0
	residualControl	
	p	1e-3
	U	1e-4
	k, epsilon	1e-3
	pRefCell	0
relaxationFactors	pRefValue	0
	p	0.4
	U, k, epsilon	0.6

The last parameters in the fvSolution dictionary are the linear solvers as shown in Table A-8. The linear solvers are used for each discretized equation and representations the method of number-crunching operations to solve the set of linear equations in addition the application solver which is the description of the problem in a set of equations and algorithms.

OPENFOAM offers several options for these linear solvers. For a quick solution it is possible to apply the Generalised geometric-algebraic multi-grid (GAMG) solver which uses a coarser mesh to make an initial guess. This method also includes a smooth solver which can also be applied separately. However, as the results of these methods are used not fully accurate due to these smoothening functions, the Preconditioned (bi-)conjugate gradient (PCG and PBiCG) solvers are used for all variables. These solvers are practically identical, but PCG is symmetric and PBiCG is not. A preconditioner is used to generate a system that converges much faster than the original one. Used are the so called Simplified diagonal-based incomplete Cholesky (DIC) preconditioner for the symmetric pressure solver and the simplified diagonal based incomplete LU (DILU) preconditioner for the asymmetric variables.

Table A-8: Linear solvers in the fvSolution dictionary

Sub-dictionary	Sub-sub-dictionary	Keyword	Entry
Solvers	p	solver	PCG
		preconditioner	DIC
		tolerance	1e-5
		relTol	1e-2
	U, k, epsilon	solver	PBiCG
		preconditioner	DILU
		tolerance	1e-5
		relTol	1e-2

Appendix G IMPLEMENTATION AS BOUNDARY CONDITION

A different method than the explained actuator disc, would be to not add the forces as momentum within the grid, but to apply it as a velocity boundary condition. This would mean that the duct is split into two parts as shown in Figure A-7. On the boundary condition at the right (marked with 1), the desired velocity shape distribution can be plotted. A boundary condition as a function can be added with the use of the OPENFOAM contrib *swak4Foam*. At the other new boundary condition (boundary 2) the inflow to the propeller should be implemented, this can be done as a uniform velocity.

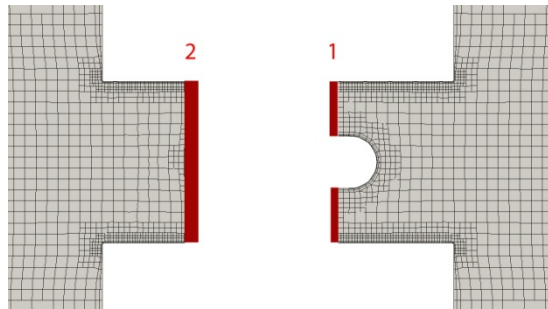
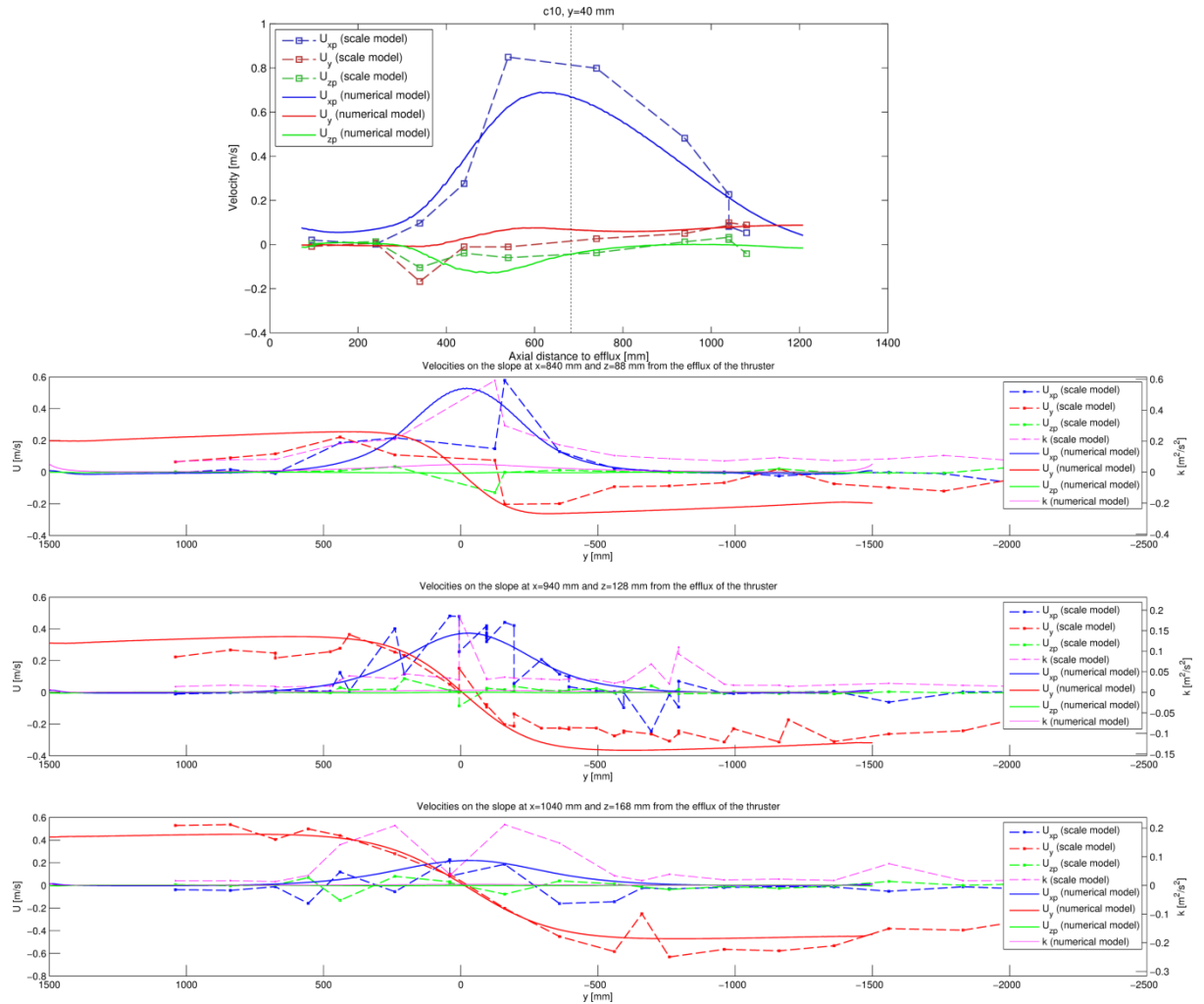


Figure A-7: Implementation of the velocities as a boundary conditions at 1 and 2

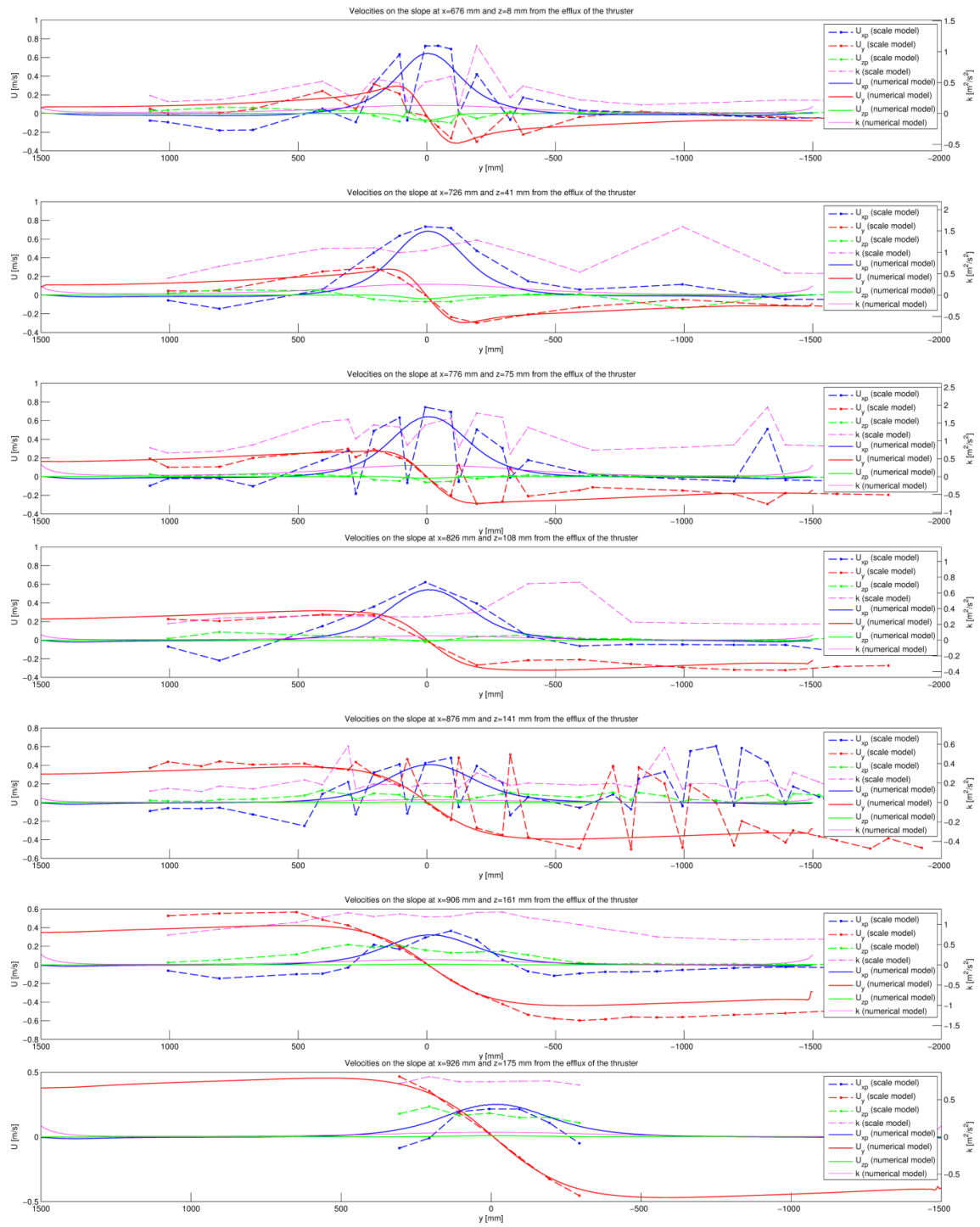
Appendix H FULL CALIBRATION FIGURES

This appendix shows an overview of the comparison of the numerical model for many of the scenarios that Van Doorn measured. In contrast to the figures in Chapter 4, these figures are not dimensionless and, besides showing the similarities between the numerical and physical scale model, it serves as an overview of the measurement data for further research.

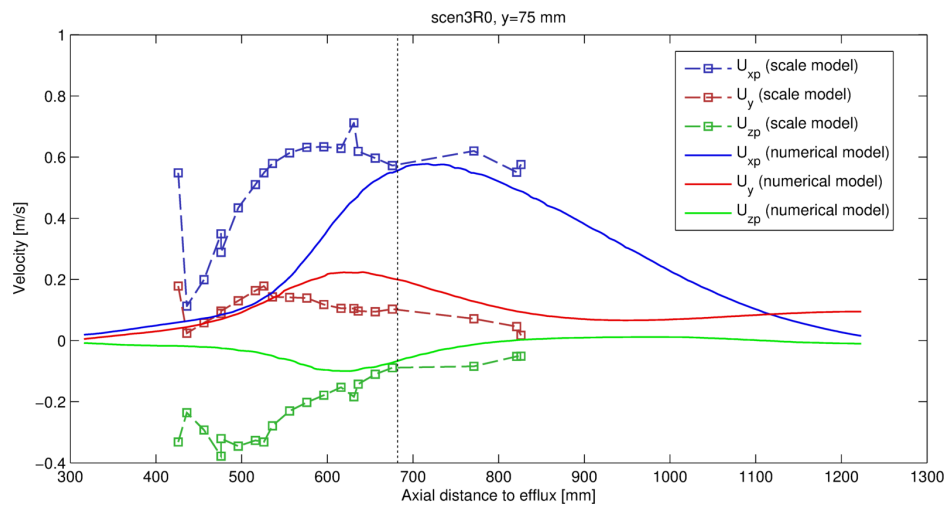
SCENARIO 1



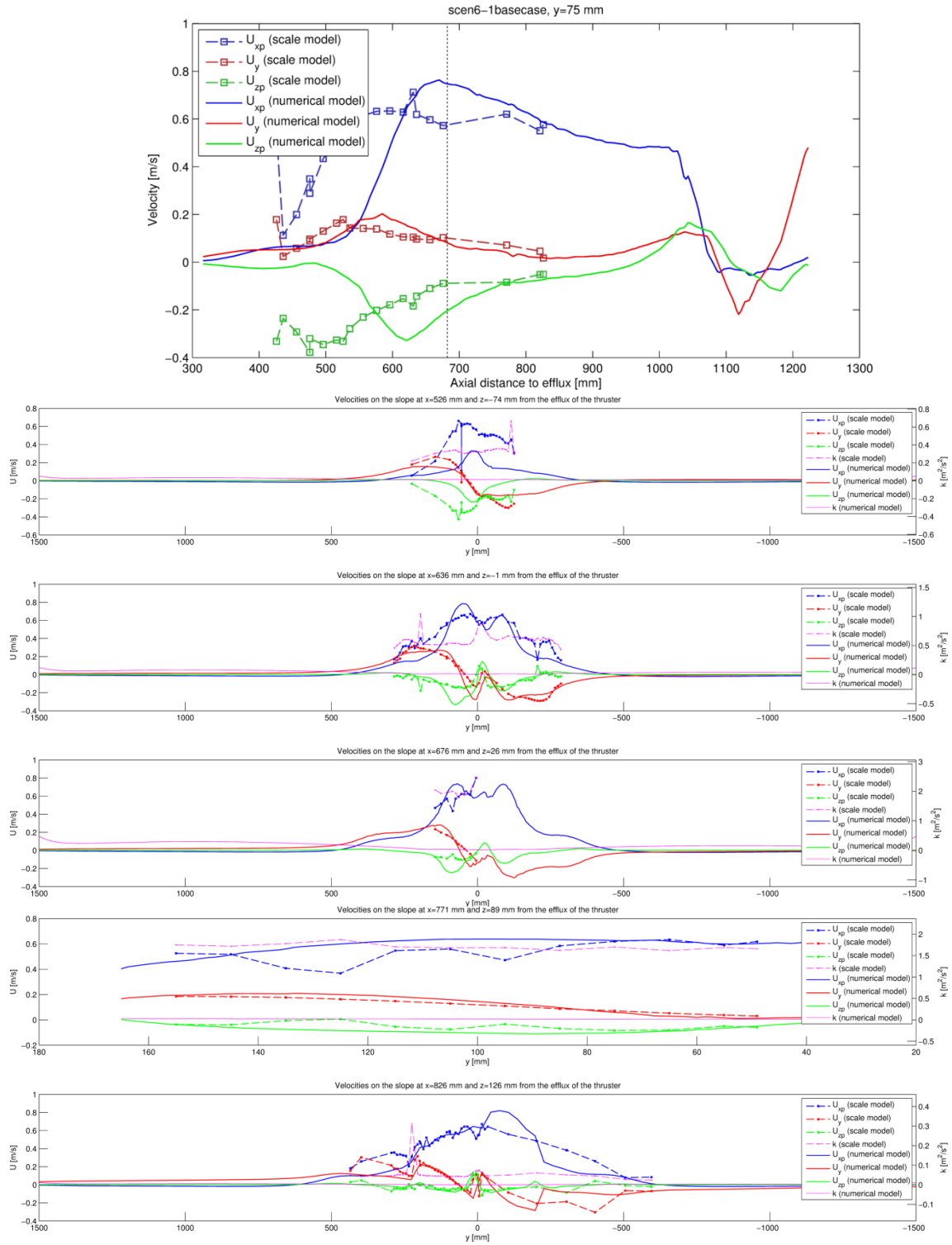
SCENARIO 2



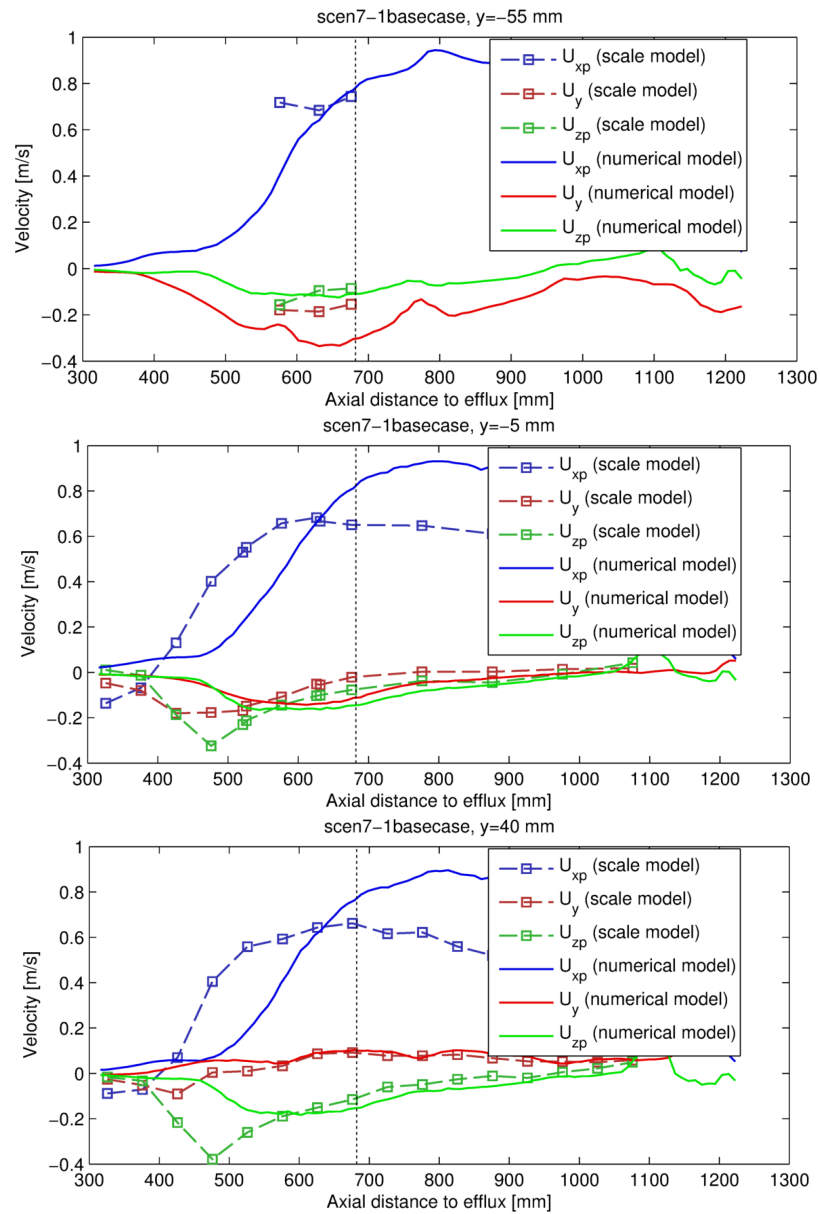
SCENARIO 3



SCENARIO 6



SCENARIO 7



Appendix I RESULTS AT PILES INCLUDING TURBULENCE

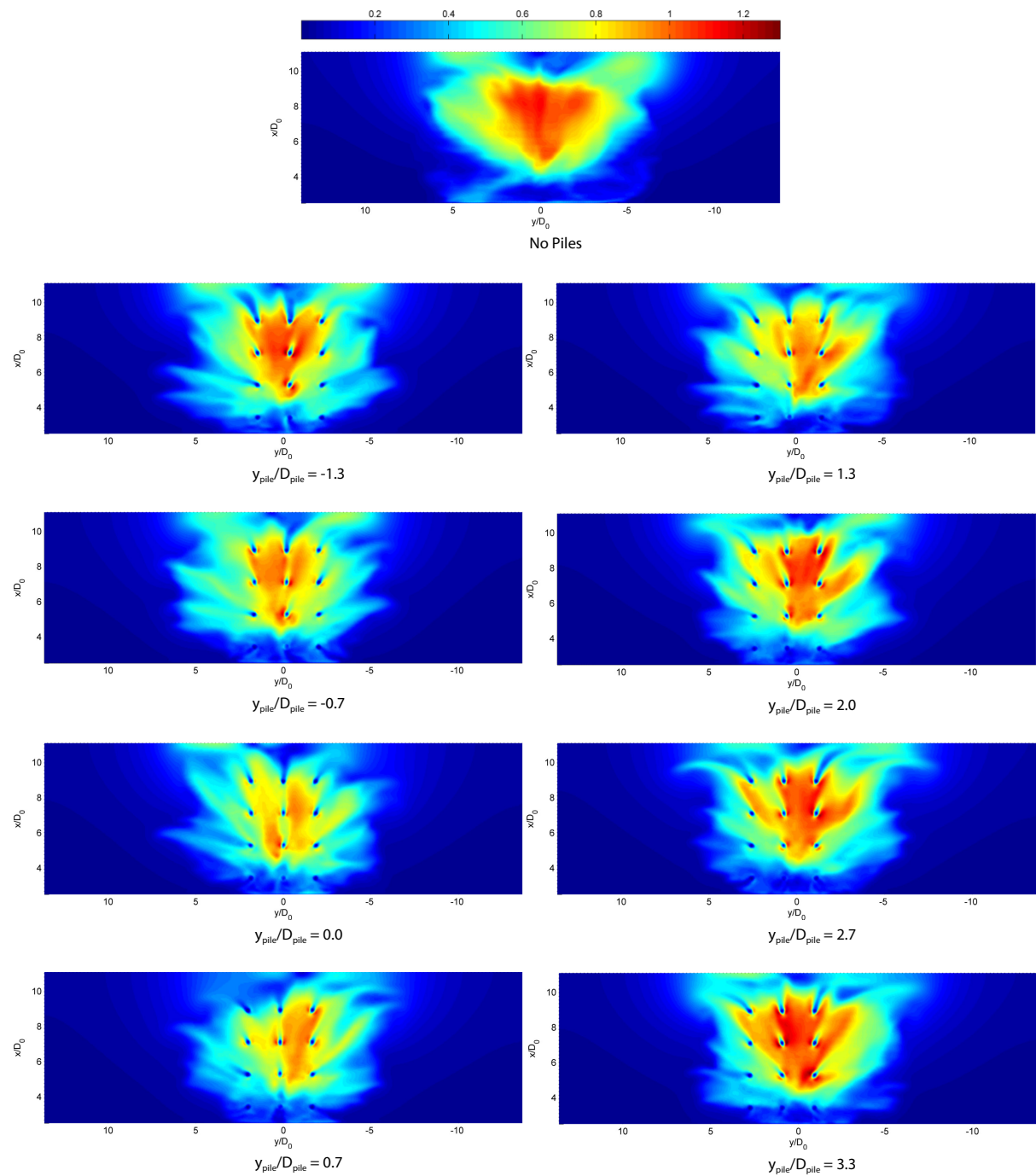


Figure A-8: Dimensionless mean velocities U/U_0 increased with three times the dimensionless turbulent velocities. Shown for different alignments of the piles to the bow thruster. The thruster axis is located at $y/D_0 = 0$ and intersects the slope at $x/D_0 = 6.2$.

Table A-9: Maximum mean velocities in the numerical for different alignments of the bow thruster to the piles

y/D_{pile}	U_m/U_0		$(U_m+3k)/U_0$	
	Field	Piles	Field	Piles
No piles	0.55	-	1.13	-
-1.3	0.57	0.75	1.10	1.40
-0.7	0.58	0.71	0.96	1.33
0.0	0.50	0.65	0.96	1.24
0.7	0.51	0.59	0.96	1.24
1.3	0.54	0.72	1.03	1.26
2.0	0.59	0.67	1.08	1.24
2.7	0.56	0.68	1.08	1.30
3.3	0.53	0.75	1.18	1.49