Numerical analysis of Bow Tunnel Thruster Performance

Anoop Mohan
Numerical Analysis of Bow Tunnel Thruster Performance

Master of Science Thesis

By

Anoop Mohan

in partial fulfillment of the requirements for the degree of

Master of Science

in Marine Technology – Track Science – Specialization Ship Hydromechanics (MT – SC - SH)

at the Delft University of Technology, Faculty of Mechanical, Maritime and Materials Engineering to be defended publicly on Tuesday April 4, 2017 at 1:30 pm.

Student number: 1542400
Thesis committee: Prof. dr. ir. R. H. M. Huijsmans TU Delft
                           Prof. dr. ir. T. J. C. van Terwisga TU Delft
                           Dr. ir. M. Godjevac TU Delft
                           Dr. ir. E. A. Munts Royal IHC
Abstract

A previous work at Royal IHC, following the observation from sea trials of trailing suction hopper dredgers (TSHD) that the turning performance of bow tunnel thrusters significantly reduced at slow forward speeds, studied the flow behaviour and performance of bow tunnel thrusters at slow forward speeds using Computational Fluid Dynamics (CFD). Simulations were performed, on model scale, first with a simplified wedge model of a containership and then with a simplified wedge model of a TSHD with the thruster modelled by an actuator disk. The numerical results deviated significantly from experimental results from literature. The present work aims to investigate this deviation. Simulations are also performed with a full thruster unit to compare with the actuator disk approach. Also, possible improvement in thruster effectivity is explored by altering the tunnel geometry of the TSHD wedge.

First, a grid convergence study is performed on the containership wedge. Next, simulations are carried out for a range of forward speeds - for selected thrusts for the actuator disk approach, and corresponding rpm’s for those thrusts for a full thruster unit. In case of the actuator disk approach, the numerical results show a steady increase of transverse force ($F_y$) with increasing forward speed, contrary to observations in sea trials. The circulation around the wedge, resulting from interaction of the jet from the tunnel and the cross-flow due to the forward speed, results in the wedge to behave as a slender body in a flow. This results in a transverse force, analogous to Kutta-Joukowski lift, that results in increased $F_y$. A full thruster unit in place of the actuator disk result in a different flow behaviour and variation of $F_y$ with forward speed. In this case, an initial decline in $F_y$ is observed before it increases monotonically. Comparison with experimental result of the containership wedge and numerical results by both actuator disk and thruster unit show considerable deviation. The transverse force due to the circulation around the hull presents a plausible explanation.

The conclusions from the grid convergence study for the containership model are utilized in meshing the TSHD wedge. Simulations performed for a range of forward speeds, first with actuator disk approximation, indicated lowered thruster performance as was observed in sea trials. The pressure contours over the hull imply that the resultant of the forces around the inlet – due to low pressure caused by high velocities of the flow into the tunnel – and those around the outlet – due to low pressure caused by high velocities of outflow – is such that there is a net thrust deduction. Simulations with thruster unit also show lowered thruster effectivity.

The tunnel of the TSHD wedge is then given a forward bend of 45° at the tunnel inlet and outlet in order to explore the possibility of improving thruster performance. It is expected that the changed geometry will result in better inflow at tunnel inlet, and the action of the cross-flow on the jet will result in a ‘straightened’ jet with lesser interaction with the hull compared to a jet from a straight tunnel. The CFD results, with actuator disk approximation, indicate that thruster effectivity improves in case of higher forward speeds and reduces for lower speeds. It is advisable to perform the study with smaller tunnel bending angles such that the jet-flow interaction is favourable for the entire range of speeds.
**Acknowledgments**

This thesis is the culmination of an year of research - initially at IHC MTI, Delft and then for the most part at Royal IHC, Kinderdijk – which I can only describe as a fruitful learning experience.

I am indebted to Royal IHC for the opportunity and the facilities for my MSc thesis. Special thanks are due to León Seijbel and Edwin Munts who selected me for the position. Edwin also guided me throughout the project with great patience, dedication and attention to detail. He helped me define the goals of the project and familiarise with ANSYS ICEM CFD and CFX. The many discussions I had with Edwin, and his feedback and overall support were immensely helpful. He also showed me a work ethic which I find worth emulating. Many thanks to my colleagues at Delft including Nikos, Mark and Vivek with whom I share a friendly camaraderie. At Kinderdijk, Jeroen, Bram, Remy, Alex, Leonard, Ron, and David contributed to a pleasant working environment. The daily trips with Jeroen saved me a lot of time, effort, and of course, money.

I am grateful to Professors Tom van Terwisga and René Huijsmans for the guidance and support they have provided throughout the thesis work, and for providing valuable insights into hydromechanics. Their advices were instrumental in enhancing the quality of the thesis. The feedback I received on presenting my results was particularly beneficial. I gratefully acknowledge Dr. Milinko Godjevac for being part of my thesis committee at a short notice. Lourdes Gallasategui, my academic counsellor at 3mE, has been a constant source of encouragement as I faced quite a few personal hurdles in my academic journey. John Stals, the Central Student Counsellor, also provided me with immense support. My sincere gratitude to Peter Naaijen for his time and effort when I was starting out in Delft. I am also grateful to Dr. Arie Romeijn, the coordinator of my track, who has been patient and accommodating.

To my friends Jiya, Ebin, Pradeep and Mo, I am forever in your debt for helping me endure when the going got tough. To Aravind, Ashkar, Anupa, and my wonderful friends in Aachen, I am grateful for giving me emotional refuge and moral support without my ever having to ask. Many thanks are due to Ratnakar Gadi and Tommy Olsen for those thoroughly enjoyable and educative group assignments.

Finally, I owe much to my parents and brother who have been unwavering in their support. They have always shared my dreams and never failed to encourage my endeavours to bring them to fruition. For this, and for everything else, I cannot thank them enough.
Contents

Abstract ................................................................................................................................. v
Acknowledgments ................................................................................................................ vii
List of Figures ..................................................................................................................... xi
List of Tables ....................................................................................................................... xiii
Nomenclature ...................................................................................................................... xv

1 Introduction ..................................................................................................................... 1
   1.1 Summary of the previous work ................................................................................. 1
   1.2 Objectives................................................................................................................... 2
   1.3 Problem approach .................................................................................................... 3
   1.4 Tools .......................................................................................................................... 3
   1.5 Outline of the report ............................................................................................... 3

2 Numerical Formulation .................................................................................................. 3
   2.1 Concepts Underlying CFD ....................................................................................... 3
      2.1.1 Governing Equations ....................................................................................... 3
      2.1.2 Strategy of CFD ............................................................................................... 3
      2.1.3 Finite Volume Method ...................................................................................... 4
      2.1.4 Modeling of Turbulence ................................................................................... 5

3 CFD study of the Nienhuis wedge ................................................................................ 7
   3.1 Nienhuis wedge with Actuator disk ....................................................................... 7
      3.1.1 Computational model ....................................................................................... 7
      3.1.2 Grid convergence study .................................................................................. 11
   3.2 Investigations with Actuator Disk ........................................................................ 16
      3.2.1 Computational Results ................................................................................... 16
   3.3 Isolated Thruster in a tunnel ................................................................................... 29
      3.3.1 Computational model ..................................................................................... 30
      3.3.2 Computational results .................................................................................... 31
   3.4 Nienhuis wedge with a thruster unit .................................................................... 33
      3.4.1 Thruster unit with 307 rpm ............................................................................. 33
      3.4.2 Thruster unit vs Actuator disk with delivered thrust of 10 N ...................... 35
      3.4.3 Comparison with experimental values ......................................................... 37
4 CFD study of the Hopper wedge .......................................................... 39
  4.1 Simulations with the Hopper wedge .............................................. 40
    4.1.1 Computational Model .......................................................... 40
    4.1.2 Results .............................................................................. 41
5 Conclusions and Recommendations .................................................. 61
  5.1 Conclusions .............................................................................. 61
    5.1.1 Conclusions for the Nienhuis wedge ........................................ 61
    5.1.2 Conclusions for the Hopper wedge .......................................... 62
  5.2 Recommendations ..................................................................... 62
Appendix A. SST-Menter turbulence model equations ............................. 65
Appendix B. Dimensional analysis of thrust .......................................... 67
Appendix C. Numerical uncertainty estimation ...................................... 69
Bibliography ..................................................................................... 71
List of Figures

Figure 1.1 Bow tunnel thruster with protection grid. Picture adapted from [4] ........................................ 1
Figure 1.2: SAABB cross section series used in the study. From [2] ....................................................... 2
Figure 2.1 Illustration of cell-centered (a) and vertex-centered (b) type control volume constructions. Adapted from [25] ................................................................. 4
Figure 3.1 Nienhuis wedge .................................................................................................................. 8
Figure 3.2 Overview of the computational domain .............................................................................. 9
Figure 3.3 Actuator disk in the tunnel .................................................................................................. 10
Figure 3.4 Geometry of the actuator disk ............................................................................................ 10
Figure 3.5 Overview of boundary conditions ...................................................................................... 11
Figure 3.6 Lack of grid convergence for hull with sharp tunnel ends .................................................. 12
Figure 3.7 Altered hull geometry at tunnel ends ................................................................................... 13
Figure 3.8 The grids used in the grid study .......................................................................................... 15
Figure 3.9 Results of grid study using the method by Eça and Hoekstra [11]. Non-weighted (a) and weighted (b) approaches ................................................................. 16
Figure 3.10 Transverse force $F_T$ vs forward speed for $T = 1.78$ N .................................................. 17
Figure 3.11 $V_{ship} = 0.015$ m/s, $T = 1.78$ N ...................................................................................... 19
Figure 3.12 $V_{ship} = 0.1$ m/s, $T = 1.78$ N ......................................................................................... 20
Figure 3.13 $V_{ship} = 0.2$ m/s, $T = 1.78$ N ......................................................................................... 21
Figure 3.14 $V_{ship} = 0.3$ m/s, $T = 1.78$ N........................................................................................ 22
Figure 3.15 $V_{ship} = 0.4$ m/s, $T = 1.78$ N ......................................................................................... 23
Figure 3.16 Parts of the hull to observe different components of transverse force ......................... 23
Figure 3.17 $C_m$ plot from Karlikov and Sholomovich ........................................................................ 25
Figure 3.18 $C_m$ plot for $T = 1.78$ N .................................................................................................. 26
Figure 3.19 $V_{ship} = 0.4$ m/s, $T = 10$ N ............................................................................................ 29
Figure 3.20 The thruster unit .............................................................................................................. 29
Figure 3.21 Isolated thruster in a tunnel: Computational domain ....................................................... 30
Figure 3.22 Interfaces between rotating and stationary domains ....................................................... 31
Figure 3.23 Grid on the propeller ....................................................................................................... 31
Figure 3.24 Flow field for $n = 504$ rpm ............................................................................................ 32
Figure 3.25 The thruster in the bow tunnel .......................................................................................... 33
Figure 3.26 $C_m$ plots for 307 rpm ...................................................................................................... 34
Figure 3.27 Velocity magnitude distribution, 488 rpm, $V_{ship} = 0.4$ m/s ....................................... 35
Figure 3.28 Velocity magnitude distribution for actuator disk, $T = 10$ N, $V_{ship} = 0.4$ m/s ....... 36
Figure 3.29 $C_m$ plots for actuator disk and thruster unit cases for delivered thrust = 10 N ...... 37
Figure 4.1 Hopper Wedge with straight tunnel .................................................................................. 39
Figure 4.2 Hopper wedge with ($45^\circ$) bent tunnel ............................................................................ 41
Figure 4.3 Hull split into parts for analysis of forces .......................................................................... 42
Figure 4.4 $V_{ship} = 0.1$ m/s, $T = 10$ N .............................................................................................. 43
Figure 4.5 $V_{ship} = 0.2$ m/s, $T = 10$ N .............................................................................................. 44
Figure 4.6 $V_{ship} = 0.3$ m/s, $T = 10$ N .............................................................................................. 46
Figure 4.7 $V_{ship} = 0.4$ m/s, $T = 10$ N .............................................................................................. 48
Figure 4.8 $V_{\text{ship}} = 0.4 \text{ m/s}, 625 \text{ rpm}$ .......................................................... 51
Figure 4.9 $V_{\text{ship}} = 0.1 \text{ m/s}, T = 10 \text{ N}, \text{ bent tunnel}$ ................................................... 53
Figure 4.10 $V_{\text{ship}} = 0.2 \text{ m/s}, T = 10 \text{ N}, \text{ bent tunnel}$ ................................................... 55
Figure 4.11 $V_{\text{ship}} = 0.3 \text{ m/s}, T = 10 \text{ N}, \text{ bent tunnel}$ ................................................... 57
Figure 4.12 $V_{\text{ship}} = 0.4 \text{ m/s}, T = 10 \text{ N}, \text{ bent tunnel}$ ................................................... 59
Figure 4.13 Variation of transverse forces with forward speed for straight and bent tunnels 60
List of Tables

Table 3.1 Main particulars of the Nienhuis wedge ................................................................. 8
Table 3.2 Particulars of the actuator disk .................................................................................. 10
Table 3.3 Uncertainty estimation. Non-weighted approach ......................................................... 15
Table 3.4 Uncertainty estimation. Weighted approach ................................................................ 15
Table 3.5 Forces for $V_{ship} = 0.0015$ m/s, $T = 1.78$ N ...................................................... 24
Table 3.6 Forces for $V_{ship} = 0.1$ m/s, $T = 1.78$ N ................................................................. 24
Table 3.7 Forces for $V_{ship} = 0.2$ m/s, $T = 1.78$ N ................................................................. 24
Table 3.8 Forces for $V_{ship} = 0.3$ m/s, $T = 1.78$ N ................................................................. 24
Table 3.9 Forces for $V_{ship} = 0.4$ m/s, $T = 1.78$ N ................................................................. 24
Table 3.10 Forces for $V_{ship} = 0.1$ m/s, $T = 10$ N ................................................................. 26
Table 3.11 Forces for $V_{ship} = 0.2$ m/s, $T = 10$ N ................................................................. 27
Table 3.12 Forces for $V_{ship} = 0.3$ m/s, $T = 10$ N ................................................................. 27
Table 3.13 Forces for $V_{ship} = 0.4$ m/s, $T = 10$ N ................................................................. 27
Table 3.14 Values of rpm and thrust for thruster unit in a tunnel .................................................. 32
Table 3.15 Thrusts computed for 307 rpm and the corresponding forward speeds ...................... 34
Table 4.1 Particulars of the Hopper wedge .............................................................................. 39
Table 4.2 Forces for $V_{ship} = 0.1$ m/s ...................................................................................... 43
Table 4.3 Forces for $V_{ship} = 0.2$ m/s ...................................................................................... 45
Table 4.4 Forces for $V_{ship} = 0.3$ m/s ...................................................................................... 46
Table 4.5 Forces for $V_{ship} = 0.4$ m/s ...................................................................................... 48
Table 4.6 Forces on the hull for 625 rpm ................................................................................... 51
Table 4.7 Forces for $V_{ship} = 0.1$ m/s, bent tunnel ................................................................. 53
Table 4.8 Forces for $V_{ship} = 0.2$ m/s, bent tunnel ................................................................. 55
Table 4.9 Forces for $V_{ship} = 0.3$ m/s, bent tunnel ................................................................. 57
Table 4.10 Forces for $V_{ship} = 0.4$ m/s, bent tunnel ............................................................... 59
Table 4.11 Comparison of transverse and longitudinal forces for the two tunnel geometries 60
# Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_{tunnel}$</td>
<td>Cross-sectional area of the tunnel</td>
<td>$[m^2]$</td>
</tr>
<tr>
<td>$C_{F_{TH}}$</td>
<td>Thrust deduction factor</td>
<td>[-]</td>
</tr>
<tr>
<td>$C_{F_{Nienhuis}}$</td>
<td>Force coefficient as defined by Nienhuis</td>
<td>[-]</td>
</tr>
<tr>
<td>D</td>
<td>Diameter of the tunnel</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$F_s$</td>
<td>Safety factor in uncertainty estimation</td>
<td>[-]</td>
</tr>
<tr>
<td>$F_y$</td>
<td>Transverse force on the hull</td>
<td>$[N]$</td>
</tr>
<tr>
<td>h</td>
<td>Typical cell size in a grid</td>
<td>$[mm]$</td>
</tr>
<tr>
<td>$K_T$</td>
<td>Thrust coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>m</td>
<td>Ratio of mean jet velocity and forward ship speed</td>
<td>[-]</td>
</tr>
<tr>
<td>n</td>
<td>Revolutions per minute (rpm) of the thruster</td>
<td>[-]</td>
</tr>
<tr>
<td>T</td>
<td>Thrust</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$T_{full,scale}$</td>
<td>Thrust in case of full scale</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$T_{model}$</td>
<td>Thrust in case of model scale</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$U_\phi$</td>
<td>Uncertainty (applies to $F_y$ in this work)</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$V_{AD}$</td>
<td>Volume of the actuator disk region</td>
<td>$[m^3]$</td>
</tr>
<tr>
<td>$V_{jet}$</td>
<td>Mean jet velocity</td>
<td>$[m/s]$</td>
</tr>
<tr>
<td>$V_s$</td>
<td>Forward ship speed</td>
<td>$[m/s]$</td>
</tr>
</tbody>
</table>

# Greek Letters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha_1, \alpha_2$</td>
<td>Coefficients in the formulation for error estimation</td>
<td>[-]</td>
</tr>
<tr>
<td>$\epsilon_\phi$</td>
<td>Discretization error (applies to $F_y$ in this work)</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$\Phi_0$</td>
<td>Estimate of the exact solution (for $F_y$ in this work)</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$\Phi_i$</td>
<td>Any integral or other functional of a local flow quantity ($F_y$ in this work)</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$\Phi_{fit}$</td>
<td>The value obtained with the second order polynomial fit</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Length scale</td>
<td>[-]</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>-------</td>
<td>------------------------------</td>
<td>-------</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Standard deviation</td>
<td>[N]</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
<td>[kg/m$^3$]</td>
</tr>
<tr>
<td>$\Delta\phi$</td>
<td>Data range parameter</td>
<td>[N]</td>
</tr>
</tbody>
</table>
1 Introduction

Bow tunnel thrusters enhance manoeuvrability of vessels during mooring and while operating at slow speeds. They can be designed to provide additional station keeping power when dynamic positioning. In case of trailing hopper dredgers wherein the present work is especially relevant, the thrusters are important for slow speed manoeuvring and course keeping. Figure 1.1 shows a typical bow tunnel thruster.

Figure 1.1 Bow tunnel thruster with protection grid. Picture adapted from [4].

It was observed in a study conducted by MARIN for Royal IHC[6] that the turning performance by using the bow thrusters is significantly reduced when the dredging vessel under consideration was moving forward at a speed of 5 knots compared to when the ship was at rest. The decrease in effectivity of tunnel thrusters with increasing forward speed has been a subject of several studies[1,2,4].

1.1 Summary of the previous work

Although the performance of lateral tunnel thrusters on ships has been investigated by many researchers for decades, Computational Fluid Dynamics modelling is a relatively late entrant in the scene. Among the earliest works are those by Ridley[9] wherein an attempt was made to correlate full-scale data to model test data pertaining to conditions with forward speed and operational thruster. Beveridge[7] examined the interaction between the ambient flow of a hull and bow thruster inflow and outflow theoretically and experimentally, and concluded that it is bow-thruster outflow, not inflow, which is an important factor in bow-thruster jet interaction at forward speed. Baniela[4] studied the effectiveness of a bow tunnel thruster with slow speed ahead, describing the general reasons for the decrease in performance, and explored anti-suction tunnel as a possible design option that can improve the effectiveness. Karlikov and Sholomovich [12] studied the interaction of thruster-generated jets with the flow near the hull at low flow velocities.

Nienhuis[1] analysed the effectivity of thrusters with numerical and experimental tools. The effects of ship speed and current on the lateral forces developed by the tunnel thruster was
considered in some detail. For this purpose, model tests were performed on a wedge-shaped hull hereafter referred to as the Nienhuis wedge. The experimental results were used to validate the computations.

Motivated by the observation at Royal IHC\cite{6} that the turning ability using a bow tunnel thruster is significantly decreased in forward speed conditions, the performance of a tunnel thruster was studied for varying tunnel cross-sections using CFD by Schaap\cite{2}. The effectivity of the thruster is represented by the thrust deduction factor $C_F$ which is calculated from the transverse forces on the hull and the thrust. CFD simulations are performed for a simplified wedge model of a containership with a bow tunnel thruster, as defined by Nienhuis\cite{1}, and validated with model test results provided by Nienhuis\cite{1}. Upon validation, while qualitative validity is established, large differences are observed between the quantitative results of the model tests from \cite{1} and those of CFD simulations \cite{2}. In these simulations, the tunnel thruster was modeled by an actuator disk. Simulations are then presented for the hull geometry of interest – a hopper wedge model based on a trailing suction hopper dredger - wherein the transverse forces are obtained and compared for varying bow tunnel cross-sections. A systematic tunnel cross-section variation is derived and three different shapes (SAABB cross section series) are computed at different ship speeds: a circular cross-section (S1010A), flattened cross-section (S0610A) and a streamlined cross-section (S0602A), as shown in Figure 1.2.

![Figure 1.2: SAABB cross section series used in the study. From [2].](image)

It is observed that the thruster effectivity is improved by increased transverse force although resistance is increased as well. The study by Schaap\cite{2} forms a basis to the current work.

### 1.2 Objectives

The objective of the present work is to further investigate the deviation between the numerical and experimental results of thruster effectivity as observed in the work by Schaap\cite{2}. In particular, the effect of the actuator disk approach is investigated by including a full thruster unit in the CFD simulations. Additionally, another concept for improving thruster efficiency is explored, namely that of a forward bent tunnel inlet and outlet.
1.3 Problem approach

Simulations can be performed for Nienhuis wedge with a tunnel thruster approximated with an actuator disk in order to do a grid convergence study. The “converged” grid is then used to study tunnel thruster effectivity for a range of forward ship speeds. The actuator disk is then replaced by a thruster (including propeller, hub, and strut) and simulations performed. The latter CFD results are expected to correspond better to the available experimental results. The 2 approaches may be compared for the same forward speed-thrust combinations. The tunnel thruster effectivity for a Hopper wedge with a full thruster can then be analysed.

1.4 Tools

As opposed to the work by Schaap[2], the present work is carried out using ANSYS software. The geometry is meshed with ANSYS ICEM CFD. The resulting unstructured mesh consists of tetrahedral and prism elements. ANSYS ICEM CFD generates hybrid tetrahedral grids consisting of layers of prism elements near the boundary surfaces to better model the near-wall physics and tetrahedral elements in the interior. The simulations are then performed using ANSYS CFX, which employs vertex-based finite volume method to solve the Reynolds Averaged Navier Stokes equations.

1.5 Outline of the report

In chapter 2, the numerical formulation is presented, including the governing equations and the solution procedure. Fundamentals of Computational Fluid Dynamics, including Reynolds averaging and turbulence modeling are summarized. In chapter 3, CFD results are presented for the Nienhuis wedge operating at low forward speeds, with a functioning tunnel thruster. In the initial section, the thruster is modelled by an actuator disk in order to compare the results with those from [2]. Next, the actuator disk is replaced by a full thruster unit. Chapter 4 describes the simulations for a hull corresponding to a Trail ing Suction Hopper Dredger, referred to as Hopper wedge. As in chapter 3, the actuator disk approximation as well as a full thruster unit are used to investigate the action of tunnel thruster in forward speed conditions.
2 Numerical Formulation

A theoretical background relevant to the present work is provided. It entails an introduction to Computational Fluid Dynamics (CFD), with emphasis on finite volume method used by ANSYS CFX. An overview of Reynolds-Averaged Navier Stokes (RANS) equations, the discretization method ANSYS CFX solves, and a brief description of turbulence models are included.

2.1 Concepts Underlying CFD

CFD deals with the prediction of fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations.

2.1.1 Governing Equations

The motion of a fluid can be described completely in terms of the Navier-Stokes or momentum equations supplemented by the continuity equation (conservation of mass) and, if required, the energy equation (conservation of energy). For an incompressible fluid, the continuity equation is given as:

$$\nabla \cdot \vec{V} = 0$$  \hspace{1cm} (2.1)

and the momentum equation, as:

$$\frac{\partial \vec{V}}{\partial t} + (\vec{V} \cdot \nabla) \vec{V} = -\frac{1}{\rho} \nabla p + \nabla \tau$$  \hspace{1cm} (2.2)

where $\vec{V}$ is the velocity vector, $\rho$ is the density, $p$ the pressure and $\tau$ the stress tensor.

These equations (along with the conservation of energy equation) form a set of coupled, nonlinear partial differential equations. For most engineering problems, it is not possible to solve these equations analytically. However, approximate computer-based solutions can be obtained for the governing equations for a variety of engineering problems. This is the crux of Computational Fluid Dynamics (CFD).

2.1.2 Strategy of CFD

Broadly, the strategy of CFD is to replace the continuous problem domain with a discrete domain using a grid. In the continuous domain, each flow variable is defined at every point in the domain. In the discrete domain, each flow variable is defined only at discrete points. CFD simulations involve discretization of the partial differential equations and numerically solving the resulting algebraic equations. Several CFD techniques exist, such as Finite Difference, Finite Element, Finite Volume and Spectral methods. The main difference between the techniques are based on the discretization process and the way in which the flow variables are approximated.

Finite Difference method, for instance, describes the unknowns of a flow problem by means of point samples at the node points of a grid. Finite difference approximations of derivatives of flow variables in terms of the nodal values are generated using truncated Taylor series. The derivatives appearing in the governing equations are replaced by finite differences resulting in algebraic equations for the values of the flow variables at each grid point. Finite Element
methods use simple piecewise functions valid on elements to describe the local variations of unknown flow variables. Spectral methods approximate the unknowns by means of truncated Fourier series or series of Chebyshev polynomials.

Of the solution methods that CFD codes use, the Finite volume technique is the most well-established, and is the one used by ANSYS CFX. The method was originally developed as a special finite difference formulation.

2.1.3 Finite Volume Method

Once a mesh has been formed, the region of interest, referred to as the computational domain, is divided into small control volumes where the conservation law will be applied. This division can be done in two ways, as shown in Figure 2.1, depending on where the solution is stored. One is the cell-centered finite volume scheme where the solution is stored at the center of each cell. In this case, the cell is the control volume. The other one, used by ANSYS CFX, vertex-based finite volume scheme where control volumes are constructed around each mesh vertex. In Figure 2.1, the shaded regions represent the control volumes.

![Figure 2.1 Illustration of cell-centered (a) and vertex-centered (b) type control volume constructions. Adapted from [25]](image)

The equations are discretized and solved iteratively for each control volume. As such, approximate values of each variable at discrete points throughout the domain are obtained. A full picture of the flow behaviour is thus derived. In the finite volume approach, the integral form of the conservation equations are applied to each control volume to get discrete equations for the node. For example, the integral form of the continuity equation for steady, incompressible flow is

\[ \int_S \vec{V} \cdot \hat{n} \, dS = 0 \] (2.3)

where S refers to the surface of the control volume and \( \hat{n} \) is the outward normal at the surface. The physical implication of the equation is that the net volume flow into the control volume is zero. In vertex-based approach, the velocity (or any other flow variable, for that matter) are then described by simple piecewise polynomial functions. The integral equation is then
transformed into an algebraic equation. Similarly, discrete equations can be obtained for conservation of momentum (and energy, if required). Boundary conditions are then applied and the system of discrete equations solved iteratively to obtain a sufficiently converged solution.

2.1.4 Modeling of Turbulence

Turbulent flows are characterized by large, random fluctuations in velocity and pressure in both space and time. Though the flows are highly unsteady and contain large range of scales, the mean or average motion is usually well-defined. Most engineering problems involve turbulent flows and thus turbulence modeling is important in their CFD solutions.

**RANS equations**

Reynolds Averaged Navier Stokes (RANS) equations makes feasible the estimation and resolution of all the scales of turbulent flows. The reduced computational requirements for the RANS equations, while still significant, are orders of magnitude less than that required for the original Navier-Stokes equations. The assumption (known as the Reynolds decomposition) behind the RANS equations is that the time-dependent turbulent velocity fluctuations can be separated from the mean flow velocity. If the velocity field \( \vec{V} \), as seen in equations 2.1 through 2.3, has the components \((u, v, w)\), the Reynolds decomposition is given as

\[
  u(t) = \bar{u} + u'(t)
\]

which implies, \( \vec{V} = \bar{\vec{V}} + \vec{V}'(t) \), where \( \vec{V}' \) has the components \((u', v', w')\).

Applying this decomposition to the Navier Stokes equations results in RANS equations given as:

\[
  \nabla \bar{\vec{V}} = 0
\]

\[
  \frac{\partial \vec{V}'}{\partial t} + (\bar{\vec{V}} \cdot \nabla) \vec{V}' = -\frac{1}{\rho} \nabla p + \frac{1}{\rho} \nabla \cdot (\bar{\vec{V}} - \rho \vec{V}' \bar{\vec{V}}')
\]

This transform introduces a set of unknowns called the Reynolds stresses, given by \( \bar{\vec{V}}' \bar{\vec{V}}' \), which are functions of the velocity fluctuations. A turbulence model is required to estimate the Reynolds stresses and produce a closed system of solvable equations.

**Turbulence modeling**

Based on the turbulent-viscosity hypothesis, a number of models are used to estimate the Reynolds stresses. The zero equation models are very robust, as they are calculated algebraically from the flow variables, and do not require solving ordinary differential equations. However, these models are not very accurate. The one equation models are those in which the transport equations are solved for one quantity, the turbulent kinetic energy, \( k \). They are more accurate than the one equation models. But, they have the drawback of being incomplete since they require a length scale, a flow dependent quantity, which needs to be specified [16].

The two equation models involve transport equations being solved for two turbulence quantities. They are more complete as they do not require any flow dependent specification. \( k - \varepsilon \) and \( k - \omega \) are the most widely used two-equation models used in commercial CFD codes [26].
In case of \( k - \epsilon \), the transport equations are solved for the two quantities: the turbulent kinetic energy \( k \) and the turbulent viscous dissipation rate \( \epsilon \). It models turbulence based on how the turbulent energy is dissipated by the eddies in the form of heat. \( k - \epsilon \) model gives good results in the free stream region, has a good convergence rate and requires lesser memory than equivalent models. But \( k \) goes to zero at the walls, hence resolution is poor close to the walls and in regions of adverse pressure gradients. Hence \( k - \epsilon \) is not suitable for systems with rotating fluids, boundary layer separations, or geometries with curved surfaces.

The \( k - \omega \) model is based on a relation between the turbulent kinetic energy \( k \) and the turbulent frequency \( \omega \). \( k - \omega \) deals well with domains having adverse pressure gradients. Hence it is well suited to the near wall regions (boundary layer and viscous sub-layers). But it can have a strong influence and sensitivity on the incoming turbulence length-scale. It is also not as good in the free stream region as the \( k - \epsilon \) model. It takes up more time for convergence and more memory than the \( k - \epsilon \) model.

ANSYS CFX provides numerous common two-equation turbulence models and Reynolds–stress models. The present work employs the widely tested shear stress transport (SST) turbulence model. The SST model is as economical as the widely used \( k-\epsilon \) model, but it offers higher fidelity, especially for separated flows, providing excellent answers on a wide range of flows and near-wall mesh conditions via blending of \( k - \omega \) and \( k - \epsilon \) formulations. A description, along with equations, of the model is provided in Appendix A. The SST model also includes an automatic wall treatment for maximum accuracy in wall shear. The automatic wall treatment switches between a low-Reynolds number formulation (i.e. direct resolution of the boundary layer) at low \( y^+ \) (dimensionless wall distance) values and a wall function approach at higher \( y^+ \) values, thus allowing the user to take advantage of a fine near-wall mesh when present.
3 CFD study of the Nienhuis wedge

The CFD study will first include simulations with the Nienhuis wedge shown in Figure 3.1, with dimensions as given in Table 3.1. The thruster is initially approximated by an actuator disk, and then by a propeller including hub and strut. The computations are validated using results from literature\[12\].

For the actuator disk case, transient simulations are employed whereas in the case with the full propeller, a frozen rotor (steady state) approach is used to model the rotating propeller. Transient simulations for the latter case for selected ship speeds gave average values of solutions close to the values obtained with frozen rotor approach. Hence, steady state simulations are deemed sufficient to capture the flow field, and there is no need to resort to the time-consuming transient simulations.

3.1 Nienhuis wedge with Actuator disk

In order to perform simulations on the wedge, an optimal grid has to be obtained. A grid convergence study, as given in section 3.1.2, yields the required optimal grid which is used for the simulations thereafter. Prior to the description of the grid convergence study, the computational model is described in the next section.

3.1.1 Computational model

Domain

The computational domain is shown in Figure 3.2, its dimensions being 9L x 8L x 2.5L, where L is the ship length. The coordinate system places the origin at the bow, on the baseline. The x-axis points from bow to stern, the positive y-axis is towards starboard, and the positive z-axis points upwards. The jet from the actuator disk delivers is directed in the positive y-direction. An actuator disk is used in the simulations to model the thrust of the thruster. A body force is applied in the region occupied by the thruster as highlighted in Figure 3.5. The body force in the actuator disk region is given as

\[ F_b = \frac{T}{V_{AD}} \] (3.1)

where T is the applied thrust and V_{AD} the volume of the actuator disk. Since the body force is directed in the positive y-direction, it is implied that the thrust acts in the opposite direction. For thrust deduction, the transverse force on the hull must be obtained in the direction opposing the thrust, i.e. in the positive y-direction.

The thrust is assumed to have a uniform distribution radially over the actuator disk, with no tangential forces. The actuator disk geometry is shown in Figure 3.4 and the dimensions are as given in Table 3.2. The dimensions correspond to the propeller dimensions as employed in the experiments by Nienhuis \[1\].
3. CFD study of the Nienhuis wedge

Figure 3.1 Nienhuis wedge

<table>
<thead>
<tr>
<th>Particular</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length straight section</td>
<td>[mm]</td>
<td>1500</td>
</tr>
<tr>
<td>Length overall</td>
<td>[mm]</td>
<td>3100</td>
</tr>
<tr>
<td>Beam</td>
<td>[mm]</td>
<td>546</td>
</tr>
<tr>
<td>Draft</td>
<td>[mm]</td>
<td>508</td>
</tr>
<tr>
<td>Tunnel diameter</td>
<td>[mm]</td>
<td>150</td>
</tr>
<tr>
<td>Tunnel center z location</td>
<td>[mm]</td>
<td>152</td>
</tr>
</tbody>
</table>

Table 3.1 Main particulars of the Nienhuis wedge
Figure 3.2 Overview of the computational domain
3. CFD study of the Nienhuis wedge

Figure 3.3 Actuator disk in the tunnel

Figure 3.4 Geometry of the actuator disk

<table>
<thead>
<tr>
<th>Particular</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner radius</td>
<td>[m]</td>
<td>0.0225</td>
</tr>
<tr>
<td>Outer radius</td>
<td>[m]</td>
<td>0.075</td>
</tr>
<tr>
<td>Thickness</td>
<td>[m]</td>
<td>0.0415</td>
</tr>
<tr>
<td>Center coordinates</td>
<td>[m]</td>
<td>1.55 0.0 0.152</td>
</tr>
</tbody>
</table>

Table 3.2 Particulars of the actuator disk
3.1 Nienhuis wedge with Actuator disk

Computational approach
A transient approach is employed for computations with the propeller modelled as an actuator disk wherein unsteady RANS equations (described in section 2.1.1) are solved using ANSYS CFX. Simulations are run until a converged solution is obtained.

Figure 3.5 shows an overview of the boundary conditions on the computational domain. The hull is set as a no-slip wall, while the water surface – the top of the domain – is treated as a free slip wall. The domain and hull are assumed to be stationary and a flow with velocity $V_{\text{ship}}$ is imposed on the inlet boundary. On the outlet boundary a constant pressure is prescribed. On the sides of the computational domain, opening boundary condition with entrainment – suitable when flow direction is unknown - is imposed.

![Figure 3.5 Overview of boundary conditions](image)

Discretization
Second-order upwind discretization is used in both space and time. The turbulence model uses first-order upwind discretization.

3.1.2 Grid convergence study
A grid convergence study is performed to quantify the discretization uncertainty. The method by Eça and Hoekstra[11], explained in Appendix B, is used. The four systematically refined grids considered for this purpose are given in Figure 3.8. For simulations, the thrust delivered by the actuator disk and the forward speed of the ship are obtained based on the values taken from a full scale simulation typically for a trailing suction hopper dredger, as described in [6].
The relationship between the thrust and speed in full scale and model scale can be written in terms of the length scale $\lambda = \frac{l_{\text{full scale}}}{l_{\text{model}}}$ as explained in Appendix A.

$$T_{\text{full scale}} = T_{\text{model}} \lambda^3$$  \hspace{1cm} (3.2)

$$V_{\text{full scale}} = V_{\text{model}} \sqrt{\lambda}$$  \hspace{1cm} (3.3)

Based on the dimensions of the Nienhuis wedge and that of the trailing suction hopper dredger in [6], $\lambda \approx 40$. The hopper has a length ($L_{\text{full scale}}$) of 120 [m] and the actuator disk generates a thrust ($T_{\text{full scale}}$) of 114 [kN]. Thus, a thrust $T_{\text{model}} = 1.78$ N and the forward speed $V_{\text{ship}} = 0.4$ m/s are calculated to be used in the simulations for the grid convergence study.

**Alteration of hull geometry at tunnel ends**

Initially, simulations were carried out with sharp tunnel edges at the intersection of the tunnel and the hull. In order to obtain grid convergence, three simulations on systematically refined grids were performed. Figure 3.6 shows the behaviour of $F_y$, the total lateral force on the ship, plotted against number of cells in each grid. A monotonic increase in $F_y$ is noted with increasing refinement of the grids, indicating the absence of grid independence. Further investigation suggested that the sharp tunnel edges leading to flow separation at the locations as a possible reason for the behaviour. To avoid this effect, the tunnel ends were rounded by using a fillet between the tunnel and the hull as shown in Figure 3.7. Using this geometry, another grid convergence study was carried out for $F_y$.

Convergent behaviour is obtained with the new geometry using 4 grids, shown in Figure 3.8. Uncertainty – an interval containing the exact solution with 95% coverage - is estimated using the method by Eça and Hoekstra [11] (as described in Appendix B) is given in Table 3.3 and Table 3.4, and the results given in Figure 3.9.

![Figure 3.6 Lack of grid convergence for hull with sharp tunnel ends](image-url)
Figure 3.7 Altered hull geometry at tunnel ends

(a) Mesh 1. h = 64 mm. 732,975 cells.
3. CFD study of the Nienhuis wedge

(b) Mesh 2. $h = 32$ mm. 7,741,850 cells

(c) Mesh 3. $h = 16$ mm. 16,532,590 cells.
3.1 Nienhuis wedge with Actuator disk

(d) Mesh 4. $h = 8$ mm. 17,415,165 cells.

Figure 3.8 The grids used in the grid study

<table>
<thead>
<tr>
<th>$\sigma$</th>
<th>$\Delta \phi$</th>
<th>$F_i$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0415</td>
<td>0.0967</td>
<td>3</td>
</tr>
</tbody>
</table>

Table 3.3 Uncertainty estimation. Non-weighted approach

<table>
<thead>
<tr>
<th>$h$</th>
<th>$\Phi_i$</th>
<th>$\Phi_{hi}$</th>
<th>$\Phi_i - \Phi_{hi}$</th>
<th>$\epsilon_{\phi}$</th>
<th>$U_{\phi}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh 1</td>
<td>64</td>
<td>-2.96</td>
<td>-2.96</td>
<td>-0.0023</td>
<td>0.272</td>
</tr>
<tr>
<td>Mesh 2</td>
<td>32</td>
<td>-2.77</td>
<td>-2.75</td>
<td>0.0165</td>
<td>0.082</td>
</tr>
<tr>
<td>Mesh 3</td>
<td>16</td>
<td>-2.67</td>
<td>-2.70</td>
<td>-0.0329</td>
<td>0.018</td>
</tr>
<tr>
<td>Mesh 4</td>
<td>8</td>
<td>-2.71</td>
<td>-2.69</td>
<td>0.0189</td>
<td>0.022</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>$\sigma$</th>
<th>$\Delta \phi$</th>
<th>$F_i$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0455</td>
<td>0.0967</td>
<td>3</td>
</tr>
</tbody>
</table>

Table 3.4 Uncertainty estimation. Weighted approach

<table>
<thead>
<tr>
<th>$h$</th>
<th>$\Phi_i$</th>
<th>$\Phi_{hi}$</th>
<th>$\Phi_i - \Phi_{hi}$</th>
<th>$\epsilon_{\phi}$</th>
<th>$U_{\phi}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh 1</td>
<td>64</td>
<td>0.067</td>
<td>-2.960</td>
<td>-2.969</td>
<td>-0.009</td>
</tr>
<tr>
<td>Mesh 2</td>
<td>32</td>
<td>0.133</td>
<td>-2.770</td>
<td>-2.739</td>
<td>0.031</td>
</tr>
<tr>
<td>Mesh 3</td>
<td>16</td>
<td>0.267</td>
<td>-2.670</td>
<td>-2.701</td>
<td>-0.031</td>
</tr>
<tr>
<td>Mesh 4</td>
<td>8</td>
<td>0.533</td>
<td>-2.710</td>
<td>-2.701</td>
<td>0.009</td>
</tr>
</tbody>
</table>
Non-weighted approach (Table 3.3, Figure 3.9(a)) and weighted approach (Table 3.4, Figure 3.9(b)) are explained in Appendix B. It is noted that the standard deviation $\sigma$ is slightly smaller for the non-weighted approach which implies the uncertainties obtained therein is relatively more reliable.

A possible reason for lack of convergence in case of original hull geometry with sharp tunnel-hull intersections is the flow separation at the sharp edges at tunnel inlet and exit. When the edges are made smooth with fillets, the boundary layer is able to follow the smooth turn in corner, thereby minimizing flow separation.

### 3.2 Investigations with Actuator Disk

After confirmation of grid convergence, the effect of forward speeds of the vessel on thruster performance was investigated. The computational model is as given in section 3.1.1, and mesh 2 is chosen as the optimally suited grid for the simulations. A constant thrust of 1.78 N, as in the previous section, is considered at first. A higher value of thrust, 10 N, is considered thereafter. The following forward speeds are considered: $V_{ship} = 0.015$ m/s, $V_{ship} = 0.1$ m/s, $V_{ship} = 0.2$ m/s, $V_{ship} = 0.3$ m/s, and $V_{ship} = 0.4$ m/s.

#### 3.2.1 Computational Results

**Thrust $T = 1.78$ N**

The flow fields obtained for the different forward speed cases are given in Figure 3.11 through Figure 3.15. The bending of the jet towards the hull can be observed as forward speed increases. The deflection of the jet is observed to cause a redistribution of pressure as it becomes oriented in the longitudinal direction. The pressure contours on the hull can be compared for the cases with forward speeds with the minimum speed (nearly zero speed) case. Further, the hull can be split into different parts, as shown in Figure 3.16. The transverse forces on the various parts are
tabulated for each forward speed case. The redistribution of lateral forces on the hull due to forward speed can thus be better described, as given in Table 3.5 through Table 3.9.

Figure 3.10 Transverse force $F_y$ vs forward speed for $T = 1.78$ N

The variation of the transverse force $F_y$ with forward speed of the wedge is shown in Figure 3.10. There is a steady increase in magnitude in $F_y$ as the forward speed $V_{ship}$ increases. As mentioned in section 3.1.1, the computed transverse force $F_y$ must be positive to indicate thrust deduction. The negative sign implies that $F_y$ is in the direction of thrust, indicating a net gain in thrust. This gain in thrust increases as forward speed increases. The distribution of pressure over the hull, for each forward speed, can be considered in an attempt to explain this observation.

A jet in the near-absence of cross-flow is seen in Figure 3.11(a) wherein it suffers a minor deflection towards the hull. As CFX did not provide sufficiently converged results for an inlet velocity of 0, therefore a small value ($V_{ship} = 0.015$ m/s) was used as the boundary condition. The corresponding pressure contours in Figure 3.11(b) and (c) show a uniform pressure over the hull except around the tunnel ends where the high velocities at the entry and exit lead to lower pressures. It is seen that $F_y$ is negative implying a gain in thrust which is contrary to experimental observation[6]. From Table 3.5, it is observed that $F_y$ is positive for the wedge, bow and stern parts, and it is the contribution on the tunnel ends that makes the net $F_y$ on the hull in the direction of the thrust. This means that the largest contribution to the total force comes from the acceleration of the flow at the inlet of the tunnel.

At the slightly higher forward speed $V_{ship} = 0.1$ m/s, as in Figure 3.12, the flow field is only slightly different from the lowest speed case. The pressure is uniform over the hull, except in the vicinity of the tunnel ends where the in- and outflow cause a lower pressure due to high
velocities. Based on Table 3.6, as in the previous case, the contribution from tunnel ends results in a net gain of thrust. The acceleration at the inlet side creates a larger force than the bending of the jet does at the outlet side of the tunnel.

At $V_{ship} = 0.2$ m/s, the deflection of the jet is more significantly visible (Figure 3.13(a)) in the flow field. From the pressure distributions in Figure 3.13(b) and (c), the pressure is largely uniform over the hull though areas of low pressure are seen to develop in the wedge part, around the tunnel ends. This is particularly significant on the outlet side. This results in the net transverse force contribution on the wedge part to decrease in magnitude. This is further confirmed in Table 3.7 which shows that the forces on bow and stern have been increasing while the contribution on the wedge part has reduced to a small value in the direction opposing thrust. For $V_{ship} = 0.3$ m/s and $V_{ship} = 0.4$ m/s, it is observed from Figure 3.14 and Figure 3.15 that the jet is significantly bent towards the hull. The contours show redistribution of pressure on the hull. As the forward speed increases, the forces on bow and stern also increase in magnitude while the net force on the wedge part decreases until it becomes negative – i.e. in the direction of thrust – at $V_{ship}$ higher than 0.2 m/s. The forces on bow and stern increase in magnitude, while always directed in the direction of the thrust, such that by $V_{ship} = 0.3$ m/s the combination of these components decide the magnitude and direction of the net $F_y$. The force on tunnel ends does not vary significantly with forward speed. The transverse force on the tunnel, which is shearing in nature, remains positive and increases with forward speed.
3.2 Investigations with Actuator Disk

(b) Hydrodynamic pressure. The inlet side (Port)

(c) Hydrodynamic pressure. The outlet side (Starboard)

Figure 3.11 $V_{ship} = 0.015$ m/s, $T = 1.78$ N

(a) Velocity magnitude distribution
3. CFD study of the Nienhuis wedge

Figure 3.12 $V_{\text{ship}} = 0.1 \text{ m/s}, T = 1.78 \text{ N}$
(b) Hydrodynamic pressure. The inlet side (Port)

(c) Hydrodynamic pressure. The outlet side (Starboard)

Figure 3.13 $V_{\text{ship}} = 0.2 \text{ m/s}, T = 1.78 \text{ N}$

(a) Velocity magnitude distribution
(b) Hydrodynamic pressure. The inlet side (Port)

(c) Hydrodynamic pressure. The outlet side (Starboard)

Figure 3.14 $V_{\text{ship}} = 0.3$ m/s, $T = 1.78$ N

(a) Velocity magnitude distribution
3.2 Investigations with Actuator Disk

(b) Hydrodynamic pressure. The inlet side (Port)

(c) Hydrodynamic pressure. The outlet side (Starboard)

Figure 3.15 $V_{\text{ship}} = 0.4$ m/s, $T = 1.78$ N

Figure 3.16 Parts of the hull to observe different components of transverse force
### Table 3.5 Forces for $V_{ship} = 0.0015$ m/s, $T = 1.78$ N

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>0.326</td>
<td>0.481</td>
<td>0.203</td>
<td>-0.786</td>
<td>0.028</td>
</tr>
<tr>
<td>Starboard</td>
<td>-0.392</td>
<td>-0.262</td>
<td>-0.252</td>
<td>0.070</td>
<td>0.041</td>
</tr>
<tr>
<td><strong>Total [N]</strong></td>
<td><strong>-0.066</strong></td>
<td><strong>0.218</strong></td>
<td><strong>-0.048</strong></td>
<td><strong>-0.716</strong></td>
<td><strong>0.069 : -0.543</strong></td>
</tr>
</tbody>
</table>

### Table 3.6 Forces for $V_{ship} = 0.1$ m/s, $T = 1.78$ N

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>-0.217</td>
<td>-0.826</td>
<td>-0.127</td>
<td>-0.815</td>
<td>0.028</td>
</tr>
<tr>
<td>Starboard</td>
<td>0.133</td>
<td>1.055</td>
<td>0.072</td>
<td>0.102</td>
<td>0.042</td>
</tr>
<tr>
<td><strong>Total [N]</strong></td>
<td><strong>-0.084</strong></td>
<td><strong>0.229</strong></td>
<td><strong>-0.055</strong></td>
<td><strong>-0.713</strong></td>
<td><strong>0.070 : -0.553</strong></td>
</tr>
</tbody>
</table>

### Table 3.7 Forces for $V_{ship} = 0.2$ m/s, $T = 1.78$ N

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>-1.777</td>
<td>-5.188</td>
<td>-1.657</td>
<td>-1.190</td>
<td>0.043</td>
</tr>
<tr>
<td>Starboard</td>
<td>1.397</td>
<td>4.979</td>
<td>0.957</td>
<td>0.462</td>
<td>0.064</td>
</tr>
<tr>
<td><strong>Total [N]</strong></td>
<td><strong>-0.380</strong></td>
<td><strong>-0.209</strong></td>
<td><strong>-0.700</strong></td>
<td><strong>-0.728</strong></td>
<td><strong>0.107 : -1.909</strong></td>
</tr>
</tbody>
</table>

### Table 3.8 Forces for $V_{ship} = 0.3$ m/s, $T = 1.78$ N

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>-3.157</td>
<td>-8.963</td>
<td>-3.080</td>
<td>-1.310</td>
<td>0.050</td>
</tr>
<tr>
<td>Starboard</td>
<td>2.631</td>
<td>8.511</td>
<td>1.783</td>
<td>0.663</td>
<td>0.074</td>
</tr>
<tr>
<td><strong>Total [N]</strong></td>
<td><strong>-0.526</strong></td>
<td><strong>-0.452</strong></td>
<td><strong>-1.297</strong></td>
<td><strong>-0.646</strong></td>
<td><strong>0.124 : -2.797</strong></td>
</tr>
</tbody>
</table>

### Table 3.9 Forces for $V_{ship} = 0.4$ m/s, $T = 1.78$ N

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>-3.157</td>
<td>-8.963</td>
<td>-3.080</td>
<td>-1.310</td>
<td>0.050</td>
</tr>
<tr>
<td>Starboard</td>
<td>2.631</td>
<td>8.511</td>
<td>1.783</td>
<td>0.663</td>
<td>0.074</td>
</tr>
<tr>
<td><strong>Total [N]</strong></td>
<td><strong>-0.526</strong></td>
<td><strong>-0.452</strong></td>
<td><strong>-1.297</strong></td>
<td><strong>-0.646</strong></td>
<td><strong>0.124 : -2.797</strong></td>
</tr>
</tbody>
</table>
### Comparison with Karlikov and Sholomovich

The authors in [12] observed that the performance of bow thrusters is hampered by the strong dependence of their effectiveness on the ship’s speed. There is an initial decrease in lateral forces on the ship, and the effectiveness is eventually restored as the speed increases. The behavior observed in [12] is given in Figure 3.17. In this figure, the coefficient

\[ C = \frac{C_y}{C_{y,0}} \]  

(3.4)

where the lateral force coefficient \( C_y \) is defined as

\[ C_y = \frac{2F_y}{\rho V_{jet}^2 A_{tunnel}} \]  

(3.5)

where \( F_y \) is the lateral force on the ship, \( V_{jet} \) the tunnel jet speed and \( A_{tunnel} \) the area of the tunnel. \( C_{y,0} \) is the force coefficient at zero forward speed, and the parameter \( m \) is the ratio of the ship speed and the velocity of the jet given by

\[ m = \frac{V_{ship}}{V_{jet}} \]  

(3.6)

![Figure 3.17 C-m plot from Karlikov and Sholomovich](image)

The results of the simulations is as shown in Figure 3.18. The CFD results obtained differ vastly from the results in literature in terms of magnitude and behaviour of \( C \). In place of a drop in value of \( C \), a monotonous increase is observed in the computed results and thrust deduction is not observed. However, it must be noted that the hull used by Karlikov and Sholomovich [12] is different from the Nienhuis wedge, and the behaviours in Figure 3.17 and Figure 3.18 need not necessarily agree.
Further, experiments [6] indicate that the performance, in case of the hopper, initially decreases with forward speed before increasing again. It is concluded that a Nienhuis wedge, with a tunnel thruster approximated by an actuator disk for thrust $T = 1.78$ N, shows a different behaviour. The Nienhuis wedge behaves as a slender body in the flow, with redistributed pressure over the hull (including the bow and stern) due to the inflow interacting with the jet from the tunnel. As a result, the wedge experiences a transverse force which manifests as an increase in the net $F_y$, and hence in the coefficient $C$.

**Thrust $T = 10$ N**

The thrust by the actuator disk is set at 10 N and simulations are performed for a range of forward speeds as in the preceding case. Figure 3.19 shows the distributions of velocity magnitude and hydrodynamic pressure on the hull for the ship speed 0.4 m/s. As in the previous case with thrust $T = 1.78$ N, the performance of the thruster does not decrease, and, instead, an increase is obtained with increasing forward ship speed. Table 3.10 – 3.13 show the forces on the different parts of the hull for $T = 10$ N for various forward speed conditions.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-0.351</td>
<td>-3.270</td>
<td>-0.220</td>
<td>-4.580</td>
<td>-0.006</td>
</tr>
<tr>
<td>Starboard</td>
<td>0.044</td>
<td>3.710</td>
<td>0.130</td>
<td>0.220</td>
<td>0.000</td>
</tr>
</tbody>
</table>

| Total [N] |  -0.307  |  0.440   |  -0.090  |  -4.360        |  -0.006    |

Table 3.10 Forces for $V_{ship} = 0.1$ m/s, $T = 10$ N
3.2 Investigations with Actuator Disk

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>0.124</td>
<td>-1.200</td>
<td>0.099</td>
<td>-4.640</td>
</tr>
<tr>
<td>Starboard</td>
<td>-0.268</td>
<td>2.510</td>
<td>-0.002</td>
<td>0.220</td>
</tr>
</tbody>
</table>

Total [N] | -0.144 | 1.310 | 0.097 | -4.420 | -0.006 : -3.163

Table 3.11 Forces for $V_{ship} = 0.2$ m/s, $T = 10$ N

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>1.680</td>
<td>2.690</td>
<td>1.090</td>
<td>-4.490</td>
</tr>
<tr>
<td>Starboard</td>
<td>-1.820</td>
<td>-2.520</td>
<td>-1.010</td>
<td>0.190</td>
</tr>
</tbody>
</table>

Total [N] | -0.140 | 0.170 | 0.080 | -4.300 | -0.006 : -4.196

Table 3.12 Forces for $V_{ship} = 0.3$ m/s, $T = 10$ N

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>4.230</td>
<td>8.850</td>
<td>2.700</td>
<td>-4.450</td>
</tr>
<tr>
<td>Starboard</td>
<td>-4.430</td>
<td>-8.840</td>
<td>-2.640</td>
<td>0.150</td>
</tr>
</tbody>
</table>

Total [N] | -0.200 | 0.010 | 0.060 | -4.300 | -0.006 : -4.436

Table 3.13 Forces for $V_{ship} = 0.4$ m/s, $T = 10$ N

In Figure 3.19(a), it is seen that the jet is only minimally influenced by the cross-flow even for a high forward speed of 0.4 m/s. This implies a uniform pressure distribution over the hull, with variations around the inlet and outlet, as seen in Figure 3.19(b) and (c). Similar to the case explained in Figure 3.11 ($T = 1.78$ N, minimum speed case), the contribution on the tunnel ends result in a force $F_y$ that is negative implying a gain in thrust.

It is seen from Table 3.10 – 3.13 that the force at tunnel ends contribute the most towards the net negative $F_y$ for all speeds considered. The magnitude of the forces at tunnel ends are also similar in the four cases, resulting in little variation in the net transverse force. It is seen the forces at the bow, wedge and stern regions vary with the forward speed. However, the magnitude of force contributions by these areas are much smaller compared to the force resulting from the acceleration of the flow at tunnel inlet.

This study further reinforces the previously obtained conclusion that numerical simulations on Nienhuis wedge does not agree with the experimental observations [6] indicating a decrease.
in performance of the thruster. Further numerical studies with the Nienhuis wedge may be performed with a full thruster unit with hub and strut, instead of the actuator disk approach.

(a) Velocity magnitude distribution

(b) Hydrodynamic pressure. The inlet side (Port)
3.3 Isolated Thruster in a tunnel

In order to investigate the influence of the use of an actuator disk to model the thrust on the numerical results, the actuator disk is replaced by a full thruster unit in the tunnel. This thruster unit is shown in Figure 3.20, consisting of the propeller blades, a hub and a strut.

In order to deliver the same thrust as that used in the actuator disk, the thruster unit’s propeller characteristics must be determined. For this purpose, the thruster unit is placed inside an isolated tunnel as shown in Figure 3.21 and a numerical simulation is carried out for a given propeller rotation speed. Then, from similarity the propeller rotation speed for the desired thrust can be calculated.

(c) Hydrodynamic pressure. The outlet side (Starboard)

Figure 3.19 $V_{ship} = 0.4$ m/s, $T = 10$ N

Figure 3.20 The thruster unit
### 3.3.1 Computational model

#### Numerical Formulation

To simulate the rotating propeller of the thruster unit, the frozen rotor approach is employed. In this approach, the propeller is fixed in a given position with respect to the hub and strut, while the rotation effects are accounted for by including source terms in the Navier-Stokes equations, viz. the centrifugal and Coriolis forces. Note that doing so, the frozen rotor model does not resolve the unsteadiness of the wakes in time resulting from the propeller blades passing the hub and strut, however, the results resemble a snapshot in time of the actual flow corresponding to the chosen propeller blade position.

#### Computational Domain

The thruster unit is simulated in the center of an isolated tunnel as shown in Figure 3.21. The rotating part of the domain containing the propeller blades is highlighted in this figure. This rotating part of the domain is connected to the remaining stationary domain at the interfaces as indicated in Figure 3.22 using the General Grid Interface (GGI) technique which is a method for connecting non-conformal and conformal mesh regions.

At the solid walls, that is the thruster unit and the tunnel walls, a no-slip conditions is applied. It can be argued that a tunnel thruster unit on a vessel operates in bollard-pull condition, as the unit is directed perpendicular to the main forward speed of the vessel. Therefore, at the inlet and outlet boundaries of the tunnel, a constant pressure boundary condition is applied, as opposed to the use of a boundary condition prescribing a given velocity. The computational grid on the thruster unit surface is shown in Figure 3.23.

![Isolated thruster in a tunnel: Computational domain](image)
3.3 Isolated Thruster in a tunnel

3.3.2 Computational results

In order to verify similarity, simulations are performed for two propeller rotation speeds, \( n = 504 \) rpm and \( n = 762 \) rpm. The resulting calculated thrust for both rotation speeds are given in Table 3.14.
Now, similarity can be checked by considering the thrust coefficient $K_T$, defined as:

$$K_T = \frac{T}{\rho n^2 D^4} \quad \text{(3.7)}$$

where $T$ [N] is the total thrust of the unit, $\rho$ [kg/m$^3$] is the fluid density and $D$ [m] the propeller blades diameter. From similarity ($K_T = \text{constant}$) it follows that from a given thrust and corresponding rotation speed ($T_1$, $n_1$), desired conditions ($T_2$, $n_2$) can be calculated from

$$\frac{T_1}{n_1^2} = \frac{T_2}{n_2^2} \quad \text{(3.8)}$$

From the numerical results it follows that

$$\frac{T_1}{n_1^2} / \frac{T_2}{n_2^2} = 0.998$$

which confirms that similarity applies in the numerical results.

In Figure 3.24, the calculated flow field for $n = 504$ rpm is shown in a cross section through the center of the tunnel. Note that the velocity magnitude is increased behind the thruster unit at the outer diameters of the tunnel, while a wake is formed behind the hub in the center of the tunnel.

<table>
<thead>
<tr>
<th>$n$ [rpm]</th>
<th>$T$ [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>504</td>
<td>4.78</td>
</tr>
<tr>
<td>762</td>
<td>10.95</td>
</tr>
</tbody>
</table>

Table 3.14 Values of rpm and thrust for thruster unit in a tunnel

Figure 3.24 Flow field for $n = 504$ rpm
3.4 Nienhuis wedge with a thruster unit

In order to investigate the influence of the actuator disk approach as employed previously, the tunnel thruster is now investigated by using a full thruster unit, using the rotation speed as calculated in the previous section.

Computational Domain

For consistency with the actuator disk simulations, the center plane of the thruster unit’s propeller blades is positioned in the center of the tunnel, see Figure 3.25.

![Figure 3.25 The thruster in the bow tunnel](image)

The computational domain and settings remain the same as described in previous sections. Similarly as in the previous simulations, Mesh 2 is chosen as an optimal grid for future simulations following the grid convergence study described in section 3.1.2. The grid containing the thruster unit is similar to that given in section 3.3.1.

3.4.1 Thruster unit with 307 rpm

A rotation speed of $n = 307$ rpm corresponds to a thrust of $T = 1.78$ N, based on the similarity as described in section 3.3.1. Simulations are performed for a range of forward speeds for this rotational speed and the efficiency of the thruster is investigated.

Comparison with Karlikov and Sholomovich

The CFD results obtained differ from the results in literature in terms of magnitude of variation in the coefficient $C$ (explained in section 3.2.1) although there is resemblance in the behaviour. In case of the obtained results, there is a slight ‘gain’ in thrust for lower values of $m$ before the decrease. The distribution of forces on the hull can be investigated for a value of $m$ that shows an increase in $C$ above 1 as well as one for which $C < 1$ and an attempt for an explanation is made. The hull can be divided into different parts for convenience of analysis as in Figure 3.16, and the force on each part may be computed. The results for 307 rpm, as shown in Figure 3.26 show a deduction in thrust for the value of parameter $m = 0.485$ or $V_{ship} = 0.2$ m/s.
The thrust of the thruster unit can also be calculated by integrating the pressure and shear stress on the thruster. It is found that the computed thrust is not just much higher than the assumed value of 1.78 N, it also varies with forward speed. These observations may be attributed to the thruster-hull interaction and the change in inflow to the thruster with forward speed. The thrusts and the corresponding speeds are as tabulated in Table 3.15.

<table>
<thead>
<tr>
<th>$V_{ship}$ [m/s]</th>
<th>Computed thrust [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.01</td>
<td>3.03</td>
</tr>
<tr>
<td>0.1</td>
<td>3.16</td>
</tr>
<tr>
<td>0.2</td>
<td>2.69</td>
</tr>
<tr>
<td>0.3</td>
<td>2.58</td>
</tr>
<tr>
<td>0.4</td>
<td>3.96</td>
</tr>
</tbody>
</table>

Table 3.15 Thrusts computed for 307 rpm and the corresponding forward speeds

It is concluded that the relation based on similarity as obtained in section 3.3.1 does not hold for a thruster in a tunnel with the vessel having a forward speed. Hence, obtaining the desired thrust for a given forward speed requires some trial and error with the propeller rotation speed. It is seen that an approximately steady thrust of 10 N can be obtained by varying the rotation speed between 488 and 500 rpm. Results obtained using these rotation speeds can be compared to the actuator disk cases with 10 N thrust. Further, C-m plots can be obtained for both the actuator disk approach and the thruster unit, for the delivered thrust of 10 N, and compared.
3.4.2 Thruster unit vs Actuator disk with delivered thrust of 10 N

The velocity distribution for the thruster unit for 488 rpm with forward speed 0.4 m/s is shown in Figure 3.27. The computed thrust for this rotation speed is 9.3 N. The velocity distribution for the actuator disk approach for the same forward speed is given in Figure 3.28.

Figure 3.27 Velocity magnitude distribution, 488 rpm, $V_{\text{ship}} = 0.4$ m/s
A stronger jet is seen in the actuator disk case compared to the one with the thruster unit although both cases have similar delivered thrusts.

The $C$-$m$ plots for both cases can be plotted and compared, as shown in Figure 3.29. It is seen from Figure 3.29 that for the thruster unit case, higher values of $m$ show a deduction in thrust, while the actuator disk approach indicate a gain in thrust for the whole range of $m$. 

Figure 3.28 Velocity magnitude distribution for actuator disk, $T = 10$ N, $V_{\text{ship}} = 0.4$ m/s
3.4 Nienhuis wedge with a thruster unit

Figure 3.29 C-m plots for actuator disk and thruster unit cases for delivered thrust = 10 N

Higher values of \( m \) imply weaker jets. Hence, for weaker jets, the thruster unit shows a deduction in thrust. On the contrary, the value of the coefficient \( C \) does not vary much with \( m \) in case of the actuator disk approach. The rotation imparted by the thruster and the wake region that forms behind the hub of the thruster unit results in a significantly different flow field.

3.4.3 Comparison with experimental values

The test results from Nienhuis[1] are used for quantitative comparison with the numerical results. Nienhuis has provided a plot of thrust deduction factor \( C_F \) against transverse speed of the hull from model tests. Since the current work deals with the hull having forward speeds, the provided values of \( C_F \) at zero speed can be used to make a quantitative comparison. Based on Schaap[2], the thrust deduction factor \( C_F \) is defined for the actuator disk approach as:

\[
C_F = \frac{F_y}{T} \quad (3.9)
\]

where \( F_y \) is the transverse force on the hull with the actuator disk delivering thrust \( T \). In case of simulations with a thruster unit, \( C_F \) is given by Schaap[2] as:

\[
C_F = \frac{T - F_y}{T} \quad (3.10)
\]

where \( T \) is the computed thrust delivered by the thruster and \( F_y \) is the transverse force on the hull.

While comparing the values quantitatively, it is to be noted that the geometry of the thruster unit used by Nienhuis is different from the one used in the current work. Nienhuis has provided
the values of $C_F$ for various rpm’s of the thruster. The value of 504 rpm is close to the value used in the current work, corresponding to a thrust of 10 N. The value of $C_F$ given by Nienhuis for 504 rpm is 0.62. The computed value of $C_F$ for the actuator disk approach is -0.24, while the value of $C_F$ obtained from simulation with the thruster unit is -0.16. The negative sign in the computed results show a gain in thrust.

The conclusion is that the thruster unit gives a value of $C_F$ closer to the value by Nienhuis although both numerical results are significantly different from the experimental value. As noted before, the thruster geometries are different for the model tests and the computations. Also, as explained in section 3.2.1, the zero forward speed condition did not provide sufficiently converged solutions. These values of $C_F$ are therefore computed at a small, non-zero forward speed. These factors must account for the variation in the results.
4 CFD study of the Hopper wedge

The Hopper wedge geometry is based on a Trailing Suction Hopper Dredger, and is given in Figure 4.1. The main particulars of the wedge are given in Table 4.1. As in the case of the Nienhuis wedge in the chapter 3, the thruster is approximated by an actuator disk as well as a propeller with hub and strut. Upon comparing the results from actuator disk approximation of the tunnel thruster and the use of a thruster unit, and considering the fact that the latter case takes more time to converge, the actuator disk approximation is given prominence. The use of the full propeller is restricted to a single case due to time constraints.

![Hopper Wedge with straight tunnel](image)

Figure 4.1 Hopper Wedge with straight tunnel

<table>
<thead>
<tr>
<th>Particular</th>
<th>Symbol</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length overall</td>
<td>L</td>
<td>[mm]</td>
<td>3100</td>
</tr>
<tr>
<td>Beam</td>
<td>B</td>
<td>[mm]</td>
<td>1506</td>
</tr>
<tr>
<td>Draft</td>
<td>T</td>
<td>[mm]</td>
<td>508</td>
</tr>
<tr>
<td>Tunnel diameter</td>
<td>D</td>
<td>[mm]</td>
<td>150</td>
</tr>
<tr>
<td>Tunnel center location above keel</td>
<td>(z_{\text{ref}})</td>
<td>[mm]</td>
<td>109.6</td>
</tr>
</tbody>
</table>

Table 4.1 Particulars of the Hopper wedge
4.1 Simulations with the Hopper wedge

The grid convergence study from section 3.1.2 yielded an optimal grid for which the simulations thereafter were carried out for the Nienhuis wedge. The conclusions of the grid convergence study – the cell sizes for local mesh refinements - are used to mesh the Hopper wedge, so that in view of time a time-consuming convergence study is not repeated.

4.1.1 Computational Model

The computational domain and approach are as in section 3.1.1. The actuator disk geometry and dimensions are also the same as in Figure 3.4 and Table 3.2, except the center coordinates. Aside from the differences in geometry between the Nienhuis and Hopper wedges, other aspects of the computational model are the same in both cases.

The actuator disk for the Hopper wedge is first set to deliver a thrust of 1.78 N, for a forward speed 0.1 m/s, as in section 3.2. However, converged results could not be obtained for this value of thrust for the Hopper wedge even after a large number of time steps. At a higher thrust of 10 N, converged results were readily obtained and it was decided that the focus will be on this value of thrust. Simulations are then performed for a range of forward ship speeds. The cases with forward ship speed $V_{ship} = 0.1 \text{ m/s}, 0.2 \text{ m/s}, 0.3 \text{ m/s} \text{ and } 0.4 \text{ m/s}$ are considered. Two types of tunnel geometries are considered. At first, the geometry given in Figure 4.1, with a straight tunnel, is used in the numerical study. The loss of performance of the thrusters, as in the experiments, is attempted to be demonstrated. The actuator disk is then replaced by a thruster unit, as in section 3.4, with an rpm to deliver a thrust of approximately 10 N. The thruster unit is employed only for the forward speed of 0.4 m/s for comparison. Next, the inlet and outlet of the tunnel are bent forward (towards the bow) as shown in Figure 4.2 wherein the angle is $45^0$ to the longitudinal axis. It is intended that the bent inlet provides a better inflow. At the outlet, the cross-flow is expected to counteract the bent jet flow and hence the low pressure on the hull associated with it. The forces are analyzed for both geometries and it is investigated whether the jet at an angle to the cross-flow improves the performance of the thruster at different forward speeds. The longitudinal (drag) force $F_x$ and the transverse force on the hull $F_y$ for straight and bent tunnel for each forward speed can be tabulated and compared, as in Table 4.11.
4.1 Simulations with the Hopper wedge

4.1.2 Results

**Straight tunnel with Actuator disk**

The velocity distributions and pressure distributions on the hull in case of the straight tunnel for ship speeds $V_{\text{ship}} = 0.1 \text{ m/s}, 0.2 \text{ m/s}, 0.3 \text{ m/s} \text{ and } 0.4 \text{ m/s}$ are given in Figure 4.4 through Figure 4.7. As in section 3.2.1, varying deflection of the jet is seen as forward speed varies. The pressure contours give an indication of the redistribution of pressure on the hull. As shown in Figure 4.3, same as the case with Nienhuis wedge in Figure 3.16, the hull can be split into parts and the contribution of each towards the net transverse force can be tabulated. Tables 4.2 - 4.5 present the force contribution of each part in case of each forward ship speed.
Figure 4.3 Hull split into parts for analysis of forces

(a) Velocity magnitude distribution
4.1 Simulations with the Hopper wedge

(b) Hydrodynamic pressure. The inlet side (Port)

(c) Hydrodynamic pressure. The outlet side (Starboard)

Figure 4.4 $V_{\text{ship}} = 0.1$ m/s, $T = 10$ N

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>-1.100</td>
<td>-1.920</td>
<td>0.001</td>
<td>-2.600</td>
<td>-0.005</td>
</tr>
<tr>
<td>Starboard</td>
<td>0.490</td>
<td>4.320</td>
<td>1.110</td>
<td>0.380</td>
<td>0.000</td>
</tr>
</tbody>
</table>
| Total [N]        | -0.610  | 2.400     | 1.111     | -2.220        | -0.005     :

Table 4.2 Forces for $V_{\text{ship}} = 0.1$ m/s
Figure 4.5 $V_{ship} = 0.2 \text{ m/s}, T = 10 \text{ N}$
4.1 Simulations with the Hopper wedge

Table 4.3 Forces for \( V_{\text{ship}} = 0.2 \) m/s

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>-3.19</td>
<td>-6.82</td>
<td>-1.08</td>
<td>-2.63</td>
<td>-0.01</td>
</tr>
<tr>
<td>Starboard</td>
<td>2.54</td>
<td>9.00</td>
<td>2.22</td>
<td>0.44</td>
<td>0.00</td>
</tr>
</tbody>
</table>
| Total [N] | -0.65   | 2.18      | 1.14      | -2.19          | -0.01      | : 0.47

(a) Velocity magnitude distribution

(b) Hydrodynamic pressure. The inlet side (Port)
(c) Hydrodynamic pressure. The outlet side (Starboard)

![Pressure Distribution](image)

**Figure 4.6** $V_{ship} = 0.3$ m/s, $T = 10$ N

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>-4.21</td>
<td>-9.19</td>
<td>-1.61</td>
<td>-2.65</td>
</tr>
<tr>
<td>Starboard</td>
<td>3.54</td>
<td>11.33</td>
<td>2.84</td>
<td>0.46</td>
</tr>
</tbody>
</table>

| Total [N] | 2.14 | 1.23 | -2.19 | -0.01 |

Table 4.4 Forces for $V_{ship} = 0.3$ m/s
4.1 Simulations with the Hopper wedge

(a) Velocity magnitude distribution
From Tables 4.2 – 4.5, it is observed that there is a loss of performance in all four ship speeds considered with a straight tunnel. The forces on the hull have positive signs which, according to sign conventions, implies that the net transverse forces oppose the thrust. This is in line with the experimental observation that a hopper suffered loss of thruster performance at forward speeds [6].
For $V_{ship} = 0.1$ m/s (Table 4.2), the major contribution to the net transverse force is from the wedge part, which has a high positive force contribution, and the tunnel ends which has a negative contribution comparable in magnitude to that of the wedge. The contribution at the stern is positive and is larger (in magnitude) compared to the negative contribution at the bow. The net transverse force is thus a small positive value, indicating a thrust deduction. Upon closer examination, it is seen that the force at the inlet (port) side of the wedge part has a negative sign is much smaller in magnitude than the positive force at the outlet (starboard) side. Also, the tunnel end at the inlet has a force with larger magnitude and negative sign when compared to the force at the outlet which has a small magnitude and positive sign. This means that the bending of the jet at the outlet (starboard) side results in a higher force on the wedge than the accelerating flow at inlet (port) side of the wedge. As for the tunnel ends, the accelerating inflow results in much higher force compared to the flow at outlet. The pressure contours in Figure 4.4 also indicates a redistribution of pressure which is consistent with the interaction of the bent jet at the starboard side of the wedge resulting in a higher magnitude of force at the region.

It is observed from Tables 4.3 – 4.5 that the distribution of forces in the different parts of the hull are similar for all ship speeds considered. The same reasoning is applicable in every case to explain the net transverse force with small magnitude and positive sign. Further, it is concluded that the transverse force $F_y$ shows a decreasing trend as forward speed increases, i.e. the performance of the thruster increases as forward speed increases.

**Straight tunnel with Thruster unit**

The actuator disk is replaced by the thruster unit and simulations carried out for a propeller rotation speed of 625 rpm, corresponding to a thrust of 10 N, and the results compared with those by the actuator disk. The forward speed of 0.4 m/s is considered. The results are shown in Figure 4.8, and the transverse forces on various parts of the hull is as given in Table 4.6.
4. CFD study of the Hopper wedge

(a) Velocity magnitude distribution

(b) Hydrodynamic pressure. The inlet side (Port)
4.1 Simulations with the Hopper wedge

(c) Hydrodynamic pressure. The outlet side (Starboard)

![Hydrodynamic pressure diagram](image)

Figure 4.8 $V_{\text{ship}} = 0.4 \text{ m/s, 625 rpm}$

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>3.00</td>
<td>-16.42</td>
<td>-10.96</td>
<td>-0.66</td>
<td>0.00</td>
</tr>
<tr>
<td>Starboard</td>
<td>-2.27</td>
<td>14.67</td>
<td>10.70</td>
<td>2.97</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Table 4.6 Forces on the hull for 625 rpm

The results with the thruster unit can be compared with those with the actuator disk, in Figure 4.7 and Table 4.5. It is seen that while both cases indicate lowered thruster performance, the deduction in thrust, equal to the total transverse force $F_y$, is much higher with the thruster unit as compared to the actuator disk approach. The velocity magnitude distribution as well the pressure contours on the hull as shown in Figure 4.7 and Figure 4.8 also show significant qualitative variation. Upon comparing the distribution of forces in case of the actuator disk approach in Table 4.5 with that with the thruster unit in Table 4.6, the major contribution towards the net transverse force in the former case occurs at the outlet (starboard) side of the wedge. In the latter case, this contribution is concentrated at the tunnel end of the outlet side.

The difference can be attributed to the rotation that the thruster adds to the flow which is absent in the actuator disk approach. The presence of the hub and strut of the thruster unit also contributes to the difference in flow field.

**Bent tunnel: 45°**

In order to investigate the possibility to improve tunnel thruster performance, the tunnel inlet and outlet are bent forward (as shown in Figure 4.2). This shape provides a more aligned inflow and, additionally, it directs the jet forward counteracting the jet bending off the outflow. It is investigated whether the resulting pressure distribution on the hull would aid or oppose the
thrust. The results are shown in Figures 4.9 through 4.12, and the forces on various parts of the hull are as given in Tables 4.7 through 4.10.

(a) Velocity magnitude distribution

(b) Hydrodynamic pressure. The inlet side (Port)
4.1 Simulations with the Hopper wedge

(c) Hydrodynamic pressure. The outlet side (Starboard)

Figure 4.9 $V_{ship} = 0.1$ m/s, $T = 10$ N, bent tunnel

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>1.010</td>
<td>3.940</td>
<td>1.170</td>
<td>-2.460</td>
<td>-2.460</td>
</tr>
<tr>
<td>Starboard</td>
<td>-1.980</td>
<td>3.230</td>
<td>-0.180</td>
<td>0.160</td>
<td>1.510</td>
</tr>
</tbody>
</table>

| Total [N]   | -0.970  | 7.170     | 0.990     | -2.300         | -0.950     |

Table 4.7 Forces for $V_{ship} = 0.1$ m/s, bent tunnel
4. CFD study of the Hopper wedge

(a) Velocity magnitude distribution

(b) Hydrodynamic pressure. The inlet side (Port)
4.1 Simulations with the Hopper wedge

(c) Hydrodynamic pressure. The outlet side (Starboard)

Figure 4.10 $V_{\text{ship}} = 0.2$ m/s, $T = 10$ N, bent tunnel

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>-2.34</td>
<td>-4.17</td>
<td>-0.69</td>
<td>-2.61</td>
<td>-2.70</td>
</tr>
<tr>
<td>Starboard</td>
<td>0.93</td>
<td>11.12</td>
<td>1.84</td>
<td>0.49</td>
<td>2.62</td>
</tr>
</tbody>
</table>

Total [N]         | -1.41   | 6.95      | 1.15      | -2.12         | -0.08 :    4.49

Table 4.8 Forces for $V_{\text{ship}} = 0.2$ m/s, bent tunnel
4. CFD study of the Hopper wedge

(a) Velocity magnitude distribution

(b) Hydrodynamic pressure. The inlet side (Port)
(c) Hydrodynamic pressure. The outlet side (Starboard)

![Image](c) Hydrodynamic pressure. The outlet side (Starboard)

Figure 4.11 $V_{ship} = 0.3$ m/s, $T = 10$ N, bent tunnel

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>-4.16</td>
<td>-8.66</td>
<td>-1.61</td>
<td>-2.69</td>
</tr>
<tr>
<td>Starboard</td>
<td>2.55</td>
<td>11.02</td>
<td>2.83</td>
<td>0.81</td>
</tr>
</tbody>
</table>

Total [N] | -1.61 | 2.36 | 1.22 | -1.88 | 0.44 : 0.53

Table 4.9 Forces for $V_{ship} = 0.3$ m/s, bent tunnel
4. CFD study of the Hopper wedge

(a) Velocity magnitude distribution

(b) Hydrodynamic pressure. The inlet side (Port)
4.1 Simulations with the Hopper wedge

(c) Hydrodynamic pressure. The outlet side (Starboard)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port side</td>
<td>-4.55</td>
<td>-9.24</td>
<td>-1.65</td>
<td>-2.38</td>
<td>-2.88</td>
</tr>
<tr>
<td>Starboard</td>
<td>3.25</td>
<td>10.67</td>
<td>2.77</td>
<td>0.65</td>
<td>1.52</td>
</tr>
<tr>
<td>Total [N]</td>
<td>-1.30</td>
<td>1.43</td>
<td>1.12</td>
<td>-1.73</td>
<td>-1.36</td>
</tr>
</tbody>
</table>

For $V_{ship} = 0.1$ m/s and $V_{ship} = 0.2$ m/s, it can be seen from Table 4.7 and Table 4.8 respectively that a large and decisive contribution to the net (positive) transverse force $F_y$ is from the wedge part. Since the cross-flow is small in magnitude, the resulting jet is at an angle to the longitudinal. This results in high pressures forward of the outlet. The transverse force on the hull, $F_y$, has a high positive value indicating high thrust deduction.

In case of $V_{ship} = 0.3$ m/s, the relatively low, though positive, value of the transverse force on the hull indicates that the bent tunnel is advantageous only at higher forward speeds. This is further clear in the case $V_{ship} = 0.4$ m/s, wherein $F_y$ is negative which implies a gain in thrust instead of deduction. From Table 4.9 and Table 4.10, it is seen that the forces at the outlet (starboard) side of the wedge remain large in magnitude and with positive sign. So, it is the variation of the forces at the inlet (port) side of the wedge that is decisive. The larger forward speed results in a better inflow into the tunnel, resulting in lower pressure forward of the inlet. This results in a force contribution with negative sign thus decreasing counteracting the force at the outlet side. Hence, with increasing forward speed, the net transverse force decreases.

The straight and bent tunnel can be compared in terms of the transverse force $F_y$ as shown in Figure 4.13 and Table 4.11. There is a large difference in the variation of the transverse forces as seen in Figure 4.13. There is an interesting observation that at 0.3 m/s, the forces for...
both tunnel geometries are nearly the same. Further investigation with smaller bending angles is needed to know whether an optimal angle can be obtained at which the deduction in thrust is minimal for the whole range of speeds.

Figure 4.13 Variation of transverse forces with forward speed for straight and bent tunnels

<table>
<thead>
<tr>
<th>$V_{\text{ship}}$ [m/s]</th>
<th>$F_y$ [N] Straight tunnel</th>
<th>Tunnel at 45$^\circ$</th>
<th>$F_x$ [N] Straight tunnel</th>
<th>Tunnel at 45$^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>0.67</td>
<td>3.94</td>
<td>12.49</td>
<td>20.91</td>
</tr>
<tr>
<td>0.2</td>
<td>0.47</td>
<td>4.49</td>
<td>10.32</td>
<td>19.53</td>
</tr>
<tr>
<td>0.3</td>
<td>0.5</td>
<td>0.53</td>
<td>9.93</td>
<td>19.44</td>
</tr>
<tr>
<td>0.4</td>
<td>0.23</td>
<td>-1.84</td>
<td>11.01</td>
<td>16.55</td>
</tr>
</tbody>
</table>

Table 4.11 Comparison of transverse and longitudinal forces for the two tunnel geometries

From Table 4.11, another factor which gains significance is the longitudinal (drag) force, $F_x$. A high increase in $F_x$ is observed for each ship speed for the bent tunnel. It almost doubles in case of the bent tunnel in two smaller forward speed cases. The explanation must lie in the angle the jet makes with the longitudinal which results in a component of the thrust along the ship length, in the direction of the flow (i.e. positive x direction), which increases the net $F_x$. 
5 Conclusions and Recommendations

In this work, a CFD study of bow tunnel thruster effectivity is performed using the general purpose finite volume based solver ANSYS CFX. The tunnel thruster operates with the vessel in slow forward speed conditions. The thruster is modelled by an actuator disk as well as a full thruster unit. The Nienhuis wedge, based on a containership, and the Hopper wedge, based on a trailing suction hopper dredger, are used in the study. Simulations are carried out for different thrusts, of the actuator disk, and rpm’s, for the thruster unit, for varying forward ship speeds and the effectivity of the thruster is studied. In addition, the tunnel geometry of the Hopper wedge is altered by giving a forward bend of 45° at the tunnel inlet and outlet and possible improvement in thruster performance is investigated.

5.1 Conclusions

The conclusions are provided for studies with the Nienhuis wedge and those with the Hopper wedge.

5.1.1 Conclusions for the Nienhuis wedge

It is shown by an initial grid convergence study, which did not result in a converged grid, that the sharp tunnel ends needed to be provided with fillets. With the sharp ends removed, the occurrence of flow separation at the location is minimised. A grid study then performed on the wedge resulted in an optimal grid to be used for further simulations.

In case of simulations with the actuator disk, the transverse force on the hull increases with increasing forward speed and indicated an increase in thruster performance. It is seen that the wedge behaves as a slender body in a flow, with the jet from the tunnel interacting with the cross-flow due to forward speed, resulting in a transverse force similar to Kutta-Joukowski lift. As such, there is a net transverse force in the direction of the thrust resulting in a gain in thruster effectivity. When a full thruster unit is used in place of the actuator disk, different the flow behaviour as well as the variation of the thruster performance with forward speed result. The rotation added to the flow by the modelled rotating thruster is one possible reason for the difference from the outcome of an actuator disk. Another reason is the inclusion of the hub and strut of the thruster unit that changes the geometry. Also, the thrust delivered is a function of both the rpm and the inflow to the thruster (which, in turn, depends on the forward speed). Since there is no straightforward relation to obtain the rpm required for a desired thrust at a given forward speed, the rpm corresponding to approximately the same thrust as desired will have to be arrived at by trial and error.

Upon comparison, the numerical results from both actuator disk approach and thruster unit differ significantly from experimental results by Nienhuis. A possible explanation lies in the fact that the hull used by Nienhuis was only a wedge, while a stern and a bow were attached to the wedge for stability in the numerical simulations. The numerical results with the thruster unit are closer to the experimental values than those with the actuator disk approach.
5.1.2 Conclusions for the Hopper wedge

In case of the Hopper wedge, for both actuator disk and thruster unit, the numerical results show lowering of thruster performance in forward speeds as observed in sea trials at Royal IHC. The redistribution of pressure on the hull due to bending of the jet from the thruster by the cross-flow due to the forward speed towards the hull results in a force that opposes the thrust, leading to reduced thruster effectiveness. It is seen the transverse force on the hull, and hence the effectiveness, decreases with increasing forward speed. It can be concluded that the increasing bending of the jet from the tunnel towards the hull (aftwards) contributes to the thrust deduction.

The tunnel is given a forward bend of 45° at the inlet and outlet to investigate whether the bending of the jet by the cross-flow, towards the aft, can be counteracted and whether the effectiveness improved. The inflow into the tunnel is also improved because of the bend. It is concluded that the forward bend improves thruster performance for high forward speeds. For lower speeds the tunnel jet results in high velocities forward of the tunnel outlet leading to a net transverse force opposing the thrust and lowered effectivity.

5.2 Recommendations

Based on the conclusions of the current work, several potential areas for further research are identified.

Further studies with the thruster unit

The thruster unit was not used extensively in simulations, especially in case of the Hopper wedge. It was observed from comparison that the numerical results from the Nienhuis wedge with the thruster unit were closer to the with the experimental results than those from the actuator disk approach. The thruster unit can therefore be used in further simulations with more combinations forward speed and rpm.

Rotation speed, forward speed and desired thrust

It was established that the thrust delivered by the thruster depends on not just its rpm but the forward speed as well. The relation used in section 3.4, based on the definition of the thrust coefficient, does not hold for a thruster in a tunnel especially when the inflow into the tunnel depends on the forward speed of the vessel. A relation between the rpm, forward speed and the desired thrust is beneficial in comparison with actuator disk approach wherein a constant thrust is delivered. The time-consuming trial and error approach with the rpm of the thruster to obtain a thrust approximately the same as the desired value can thus be avoided.
5.2 Recommendations

Frozen rotor vs Transient approach
In the current work, frozen rotor approach in ANSYS CFX is used to model the rotating flow field in case of simulations with the thruster unit. Frozen rotor method, a steady state approach that solves RANS equations, is less accurate compared to the Transient rotor-stator approach that solves unsteady RANS equations and also available in ANSYS CFX. Frozen rotor approach was chosen mainly because of time constraints. Application of the Transient method can be of interest if future studies demand that the transient effects be accounted for.

Smaller bending angles
In case of the Hopper wedge with the bending angles at tunnel inlet and outlet, improvement in thruster performance was observed only for high forward speeds. The angle of bending in the forward direction can be reduced and the effects studied. It is possible that an optimal angle will be found at which the thruster performance improves for the whole range of forward speeds.

Quantitative comparison with data from full-scale experiments
All computations herein are performed on model scale. Also, no experimental results were available for comparison in case of the Hopper wedge. If full-scale experiments are to be performed and the results available for comparison, numerical studies can be performed on full scale and verified.
Appendix A. SST-Menter turbulence model equations

The SST k-ω turbulence model [Menter 1993] is a two-equation eddy-viscosity model which blends k-ω and k-ε formulations. The use of a k-ω formulation in the inner parts of the boundary layer makes the model suitable to resolve the boundary layer. The SST formulation switches to a k-ε behaviour in the free-stream thereby avoiding the common k-ω issue of high sensitivity to the inlet free-stream turbulence properties.

The eddy viscosity $\mu_t$ is defined using turbulent kinetic energy $K$ and the dissipation rate of the turbulent frequency $\omega$ as:

$$\mu_t = \frac{\rho K}{\max\left(\frac{\rho F_2}{a_1 \omega}, \frac{500\mu}{\rho d^2 \omega}\right)} \tag{A.1}$$

where $\rho$ is the density, $F_2$ is an auxiliary function, $\Omega$ is the absolute value of the vorticity and $a_1$ is taken as 0.31. $F_2$ is defined using wall distance $d$ as:

$$F_2 = \tanh\left(\max\left\{2 \frac{\sqrt{K}}{0.09 d \omega}, 500\frac{\mu}{\rho d^2 \omega}\right\}^2\right) \tag{A.2}$$

A blending function $F_1$ is used to blend between the k-ω and k-ε models:

$$F_1 = \tanh\left(\min\left\{\max\left(\frac{\sqrt{K}}{0.09 d \omega}, 500\frac{\mu}{\rho d^2 \omega}\right), \frac{4 \rho \sigma_{\omega} \kappa}{CD_{k\omega} d^2}\right\}^4\right) \tag{A.3}$$

where

$$CD_{k\omega} = \max\left\{\frac{2 \rho \sigma_{\omega} \phi_{k}(\kappa) \frac{\partial K}{\partial x_j} \frac{\partial \omega}{\partial x_j}}{\rho d^2 \omega}, 10^{-20}\right\} \tag{A.4}$$

The transport equations are given as:

$$\frac{\partial \rho K}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_j K - (\mu + \sigma_k \mu_t) \frac{\partial K}{\partial x_j}\right) = \tau_{ij} S_{ij} - \beta^* \rho \omega K \tag{A.5}$$

$$\frac{\partial \rho \omega}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_j \omega - (\mu + \sigma_{\omega} H_t) \frac{\partial \omega}{\partial x_j}\right) = P_{\omega} - \beta \rho \omega^2 + 2 \left(1 - F_1 \frac{\rho \sigma_{\omega}}{\omega} \frac{\partial K}{\partial x_j} \frac{\partial \omega}{\partial x_j}\right) \tag{A.6}$$

The production term $P_{\omega}$ is defined as:

$$P_{\omega} = 2\gamma p \left(S_{ij} - \frac{\omega S_{mn} \delta_{ij}}{3}\right) S_{ij} \tag{A.7}$$

The constants of the SST k-ω model are given as:

$$a_1 = 0.31 \quad \beta^* = 0.09 \quad \kappa = 0.41 \tag{A.8}$$

The coefficients of the k-ω models, $\phi_1$, and k-ε model, $\phi_2$, are blended as:

$$\phi = F_1 \phi_1 + (1 - F_1) \phi_2 \tag{A.9}$$

where $\phi = \{\beta, \gamma, \sigma_k, \sigma_{\omega}\}$. Table A.1 shows the constants.
<table>
<thead>
<tr>
<th>Coefficient</th>
<th>$\phi_1$ (k-(\omega) model)</th>
<th>$\phi_2$ (k-(\epsilon) model)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_k$ [-]</td>
<td>0.850</td>
<td>1.000</td>
</tr>
<tr>
<td>$\sigma_\omega$ [-]</td>
<td>0.500</td>
<td>0.856</td>
</tr>
<tr>
<td>$\beta$ [-]</td>
<td>0.075</td>
<td>0.083</td>
</tr>
<tr>
<td>$\gamma = \frac{\beta_1}{\beta^<em>} - \frac{\sigma_\omega k^2}{\sqrt{\beta^</em>}}$ [-]</td>
<td>0.553</td>
<td>0.440</td>
</tr>
</tbody>
</table>

Table A.1. Constants used in the SST k-\(\omega\) Menter model
Appendix B. Dimensional analysis of thrust

The length scale $\lambda$ between the (full scale) vessel and the model is given by

$$\lambda = \frac{L_{\text{full scale}}}{L_{\text{model}}}$$  \hspace{1cm} (B.1)

Froude scaling is assumed, i.e. the Froude number remains the same for both full and model scales.

$$\sqrt{\frac{V_{\text{full scale}}}{gL_{\text{full scale}}}} = \sqrt{\frac{V_{\text{model}}}{gL_{\text{model}}}},$$  \hspace{1cm} (B.2)

which gives

$$\frac{V_{\text{full scale}}}{V_{\text{model}}} = \sqrt{\frac{L_{\text{full scale}}}{L_{\text{model}}} = \sqrt{\lambda}$$  \hspace{1cm} (B.3)

$M$, $L$ and $\tau$ being the mass, length and time scales for dimensional analysis, thrust is given as

$$T = M L \tau^{-2}$$  \hspace{1cm} (B.4)

Given that $M \propto L^3$, the ratio of thrusts in full scale and model scale is given as:

$$\frac{T_{\text{full scale}}}{T_{\text{model}}} = \left(\frac{L_{\text{full scale}}}{L_{\text{model}}}\right)^3 \left(\frac{L_{\text{full scale}}}{L_{\text{model}}}\right) \left(\frac{T_{\text{full scale}}}{L_{\text{model}} \tau_{\text{model}}}\right)^{-2} = \left(\frac{L_{\text{full scale}}}{L_{\text{model}}}\right)^2 \left(\frac{V_{\text{full scale}}}{V_{\text{model}}}\right)^2 = \lambda^3$$  \hspace{1cm} (B.5)

$$T_{\text{full scale}} = T_{\text{model}} \lambda^3$$  \hspace{1cm} (B.6)
Appendix C. Numerical uncertainty estimation

Uncertainty

The solution verification procedure by Eça and Hoekstra estimates the uncertainty $U_\phi$ of a functional or local quantity $\phi$ from the data obtained from $n_g$ grids ($n_g \geq 4$). The method is based on power series expansions that neglect high-order terms and assume that $\phi$ has at least second-order finite derivatives. It is also assumed that the lowest-order schemes used in the discretization are second or first-order accurate.

$U_\phi$ is determined from an error estimate $\epsilon_\phi$ times a safety factor $F_s$. The error is estimated (in the current work) as

$$\epsilon_\phi \approx \delta_{12} = \phi_i - \phi_0 = \alpha_1 h_i + \alpha_2 h_i^2$$  \hspace{1cm} (C.1)

To assess the quality of the fit used to obtain $\epsilon_\phi$, a data range parameter is determined as

$$\Delta_\phi = \frac{(\phi_i)_{\text{max}} - (\phi_i)_{\text{min}}}{n_g-1}$$  \hspace{1cm} (C.2)

The uncertainty $U_\phi$ can be obtained from $\epsilon_\phi$ and the safety factor $F_s$ ($= 3$ in the present case) using the values of the standard deviation $\sigma$ and $\Delta_\phi$ to distinguish between “good” and “bad” error estimations:

- for $\sigma < \Delta_\phi$:
  $$U_\phi(\phi_i) = F_s \epsilon_\phi(\phi_i) + \sigma + |\phi_i - \phi_{\text{fit}}|$$  \hspace{1cm} (C.3)

- for $\sigma \geq \Delta_\phi$:
  $$U_\phi(\phi_i) = 3 \frac{\sigma}{\Delta_\phi} (\epsilon_\phi(\phi_i) + \sigma + |\phi_i - \phi_{\text{fit}}|)$$  \hspace{1cm} (C.4)

Least-squares solutions of power series expansions

Non-weighted and weighted approaches are defined such that

- non-weighted approach:
  $$w_i = 1 \text{ and } n w_i = 1, w_i \text{ being the weights.}$$

- weighted approach:
  $$w_i = \frac{1}{n_i} \frac{1}{\sum_{l=1}^{n_g} \frac{1}{h_i}}$$  \hspace{1cm} (C.5)

so that $\sum_{i=1}^{n_g} w_i = 1$.

The two-term expansion with first and second order terms is given as

$$\phi_{\text{fit}} = \phi_0 + \alpha_1 h_i + \alpha_2 h_i^2$$  \hspace{1cm} (C.6)

$\phi_0$, $\alpha_1$, and $\alpha_2$ are determined from the minimum of the function

$$S_{12}(\phi_0, \alpha_1, \alpha_2) = \sqrt{\sum_{i=1}^{n_g} w_i (\phi_i - (\phi_0 + \alpha_1 h_i + \alpha_2 h_i^2))^2}$$  \hspace{1cm} (C.7)

Setting $\frac{\partial S_{12}}{\partial \phi_0} = 0$, $\frac{\partial S_{12}}{\partial \alpha_1} = 0$, $\frac{\partial S_{12}}{\partial \alpha_2} = 0$ leads to a system of linear equations:
\[
\begin{bmatrix}
1 & \Sigma_{i=1}^{n_g} w_i h_i & \Sigma_{i=1}^{n_g} w_i h_i^2 \\
\Sigma_{i=1}^{n_g} w_i h_i & \Sigma_{i=1}^{n_g} w_i h_i^2 & \Sigma_{i=1}^{n_g} w_i h_i^3 \\
\Sigma_{i=1}^{n_g} w_i h_i^2 & \Sigma_{i=1}^{n_g} w_i h_i^3 & \Sigma_{i=1}^{n_g} w_i h_i^4
\end{bmatrix}
\begin{bmatrix}
\phi_0 \\
\alpha_1 \\
\alpha_2
\end{bmatrix}
= \begin{bmatrix}
\Sigma_{i=1}^{n_g} w_i \phi_i \\
\Sigma_{i=1}^{n_g} w_i \phi_i h_i \\
\Sigma_{i=1}^{n_g} w_i \phi_i h_i^2
\end{bmatrix}
\tag{C.8}
\]

with standard deviation given by

\[
\sigma_{12} = \sqrt{\frac{\Sigma_{i=1}^{n_g} w_i (\phi_i - (\phi_0 + \alpha_1 h_i + \alpha_2 h_i^2))^2}{n_g - 3}}
\tag{C.9}
\]
Bibliography


