Prediction of the performance of ducted propellers with BEM and hybrid RANS-BEM methods

Carlo Negrato
Prediction of the performance of ducted propellers with BEM and hybrid RANS-BEM methods

MASTER OF SCIENCE THESIS

Carlo Negrato

December 8, 2015

Supervisor:
Ir. J. Bosschers
Prof.dr.ir. T.J.C. van Terwisga

Thesis Committee:
Prof.dr.ir. T.J.C. van Terwisga
Dr.ir. H.J. de Koning Gans
Iir. R.G. Hekkenberg
Iir. J. Bosschers
Dr.ir. M.J.B.M. Pourquie
Dr.ir. J. Moulijn

Faculty of Mechanical, Maritime and Materials Engineering (3mE) - Delft University of Technology

An electronic version of this thesis is available at http://repository.tudelft.nl/.
This work is the result of a one-year stay at the Maritime Research Institute of the Netherlands (MARIN) and it is the final step for the Master of Science in Marine Technology at the Delft University of Technology. I am thankful to the people that made possible my experience at MARIN and contributed to achieve the results of this research. To begin with, I would like to thank my supervisor at MARIN Ir. Johan Bosschers for the daily supervision and the involvement in this research. The weekly meetings have always been both a great chance to get feedback and also an inspiring situation to capture some tips of his broad expertise on propellers and CFD. Secondly, I would like to express my sincere gratitude to Prof. dr. ir. T.J.C. van Terwisga who made possible my stay at MARIN; he has been supportive throughout the whole period I spent in Wageningen, giving sharp comments on the work with a friendly attitude and a nice open-mindedness for discussions. Then, I would like to acknowledge Dr. ir. Evert-Jan Foeth who supervised me in the early stage of this research and Ir. Douwe Rijpkema who helped me during the setup of the calculations. Both have also greatly contributed with questions and observations during the several progress meetings planned.

I would like to thank all the students of the studentenkamer at the R&D department at MARIN. It was a great chance to meet you all. The open-ground for discussion that was set in the office was stimulating for sharing questions and helped in solving common problems efficiently. Equally, the relaxing time out of the office was always good for chatting and chilling.

Last but surely not least, my special thank goes to my family. The endless support and love of my parents and relatives made possible to pursue my goals and face the challenges during my time in the Netherlands.
A ducted propeller configuration consists of a fixed annular nozzle surrounding the propeller. The nozzle has an airfoil shape which depends on the required performance of the system as well as the operating conditions. The flow-accelerating ducts provide a positive contribution to the thrust of the propulsor and they are used to increase the performance in heavy loading condition, such as in the cases of tug boats or often for azimuthal thrusters.

The performance of the propeller can be assessed with model tests or numerical simulations. As concerns the numerical simulations, boundary element methods (BEM) are used daily in the design stage for open propeller configurations, but the use for ducted propellers is still under development: viscous flow effects become important at the duct surface and the accuracy of the BEM method decreases. Alternatively, Reynolds averaged Navier-Stokes (RANS) simulations are possible but they require large CPU time so they cannot be used at the design stage routinely. In addition, a hybrid RANS-BEM method was developed at the Maritime Research Institute of the Netherlands (MARIN). The hybrid approach couples the viscous flow solution with the boundary element method: the propeller is not physically present in the RANS simulations, where it is substituted with a body force distribution whose strength is given by the BEM.

This research has the objective to determine the accuracy and limitations of the BEM and the hybrid RANS-BEM approaches to predict the performance of ducted propellers in open water condition. The MARIN in-house boundary element method Procal and RANS method ReFRESCO are used for two tests cases. The propeller is the same for both cases (Ka4-70 propeller) whereas the duct is different (duct 19A and duct 37). The data from model tests are used to validate the open water performance and a detailed flow analysis is carried out by looking at the pressure distributions on the propeller and on the duct as well as by looking at the contour plots of the flow variables.

As concerns the BEM method, it requires a modification of the duct geometry to provide a sharp trailing edge. An iterative scheme that automatically computes the change in geometry based on a pressure-equality condition is developed; as a result, the boundary element computations provide a 2% to 7% accuracy in the prediction of the open water efficiency for the design condition and the high loading condition. At the light loading condition, flow separation occurs at the outer duct surface, which is not modeled with the BEM approach. This affects largely the forces on the duct, and poses a limit for the applicability of the BEM method for light loading condition.

Considerable improvements are obtained with the coupled RANS-BEM approach for the first test case studied. An accuracy of 2% in the open water efficiency at the design condition is observed, with satisfactory prediction also at large advance ratios when flow separation at the duct outer surface is solved. However, for the second test case (duct 37), the propeller thrust and torque are constantly overpredicted. The limitation of the hybrid approach for this geometry is likely related to the extent of flow separation at the inner side of the duct occurring at the whole open water range for duct 37. Nonetheless, further investigations and additional validation data are required to confirm this hypothesis.

Finally, another objective of this study is to provide an insight on the flow behavior at the gap between the propeller and the nozzle. The ultimate goal is to give guidelines on the modeling of the flow in that region in a potential flow context. To do so, full RANS calculations carried out at MARIN in 2012 are analyzed. The full RANS approach refers to the expensive RANS computations where the propeller is physically present and the complete duct-propeller-hub system is solved. It observed that the gap flow is dominated by the detachment of a vortex at small chordwise position from the blade pressure side to the blade suction side. This is the so-called tip leakage vortex. An investigation of the detachment location and detachment angle of the tip leakage vortex shows that the numerical solution is in satisfactory agreement with the cavitation observations from the model tests; moreover, the analysis of the downstream development reveals a vortex-vortex interaction between the tip-leakage-vortex and the tip-vortex. Finally, a closer look at the flow velocity in the gap shows that the vortex obstructs the gap at the mid-chord position and that secondary structures are likely to develop at the blade tip, although a refinement in the mesh appears to be necessary to draw stronger conclusions on this matter.
1 Introduction 1
  1.1 Motivation ........................................... 1
  1.2 Background ........................................... 2
  1.3 Objectives ........................................... 6
  1.4 Report outline ....................................... 7

2 Theoretical background and numerical tools 9
  2.1 Governing equations .................................. 9
    2.1.1 Navier-Stokes equations .......................... 9
    2.1.2 Potential flow theory and the boundary element method ... 10
    2.1.3 RANS equations .................................. 13
  2.2 Procal .............................................. 15
    2.2.1 Wake alignment .................................. 16
    2.2.2 Gap model ..................................... 17
    2.2.3 Duct trailing edge ................................ 18
  2.3 ReFRESCO ............................................ 18
    2.3.1 Turbulence models ................................ 19
    2.3.2 Discretization ................................... 21
  2.4 Procal-ReFRESCO coupling ............................. 22
    2.4.1 Body force interpolation .......................... 23
    2.4.2 Transpiration velocity model ........................ 24
  2.5 Current capabilities of Procal and the hybrid Procal-ReFRESCO methods ........................ 24
  2.6 Grid refinement theory ................................ 25
  2.7 Analysis of propeller performance ..................... 26

3 Experimental data and full RANS reference data 29
  3.1 Introduction .......................................... 29
  3.2 Experimental data for the Ka4-70 propeller in 19A duct .................. 29
    3.2.1 Cavitation observations .......................... 32
  3.3 Full RANS computations of the Ka4-70 propeller in 19B duct ................ 32
    3.3.1 Tip leakage vortex and flow separation at the duct surface ................. 34
  3.4 Experimental data for the Ka4-70 propeller in 37 duct ...................... 35
  3.5 Full RANS computations of the Ka4-70 propeller in 37 duct .................. 35
    3.5.1 Processing of the chordwise pressure distribution on the duct ................ 37
  3.6 Summary ............................................. 37

4 Procal computations of the Ka4-70 propeller in 19A duct 39
  4.1 Introduction .......................................... 39
  4.2 Geometry and settings .................................. 39
  4.3 Trailing edge geometry ................................ 41
  4.4 Effect of hub diameter ................................ 44
  4.5 Transpiration velocity model ................................ 45
    4.5.1 Location of transpiration velocity .................. 46
    4.5.2 Boundary layer displacement thickness at the trailing edge ...................... 46
    4.5.3 Influence on the pressure distribution at the duct surface .................. 48
# List of Figures

1.1 Ducted propeller for an ATHS tug boat. Damen Shipyard, online source [1] .................................. 1
1.2 Sketch of a sectional view of an accelerating and a decelerating ducted propeller. The decomposition of the forces acting on the duct, the inflow velocity towards the propeller plane and the direction of the flow circulation induced by the duct are shown. Adapted from Oosterveld [2] ........................................ 2
1.3 Actuator disk theory for the ducted propeller configuration. The propeller is substituted with an actuator disk while the duct is retained. Adapted from Kuiper, [3] .................................................. 3
1.4 Typical curves of velocity on the propeller plane and ideal efficiency for different thrust ratio. Re-plotted from graphs in Kuiper [3] ................................................................. 4
1.5 Sketch of the detachment of the tip leakage vortex. In grey the blade suction side and the cross section of the duct are shown. Only part of the duct is shown ............................................. 5
1.6 Sketch of the streamlines of the flow in the gap between propeller and duct. Flow separation can occur at the propeller tip .............................................................. 5

2.2 Time history of axial velocity at the centerline of a turbulent jet. From Pope [5] ............................ 13
2.3 Mean axial velocity profile in a turbulent jet, normalized with its centerline value, as function of the cross-stream coordinate $x_2$ normalized by the distance $x_1$ from the nozzle. From Pope [5] ................................. 13
2.4 Geometry of the propulsor and attached wake sheets for the duct, the propeller and the duct .......................... 17
2.5 Working diagram of Procal for wetted flow computation, adapted from [6] ................................. 18
2.6 Working diagram of ReFRESCO, adapted from [7] ................................................................. 20
2.7 Iterative scheme for the RANS-BEM coupling. Adapted from [8] ............................................. 22
2.8 Graphical illustration of the recursive method for volume intersection in two dimension. Adapted from [9]. The cartesian mesh is representative for the BEM mesh the red polygon for a RANS cell .......................................................... 23
2.9 The four options for the location of transpiration velocity. The duct profile, the transpiration velocities and the location of the blade leading and trailing edge are shown ................................. 24

3.1 Front view of the setup for the open water tests for the Ka4-70 propeller in 19A duct. The shaft extends downstream and the orange tape minimizes the reflections of laser light during PIV measurements [10] ................................................................. 30
3.2 Open water diagram., measured at 900rpm and 1200rpm for the Ka4-70 in 19A duct. Adapted from [10] .................................................................................................................... 31
3.3 Blade-to-blade variation in the cavitation pattern for the tests carried out at MARIN in 2013 [10]. Blades three and four are shown; the nozzle is transparent. Operating condition here is $J=0.5$ and $\sigma=0.3$. Differences are seen in the extent of sheet and bubble cavitation. For this work, it is relevant that there is a difference in the chordwise position at which the vortex begins to cavitate at the blade tip ................................. 31
3.4 Grid distribution on the surface and on the domain for the full RANS calculations of the Ka-4-70 propeller in 19A duct. MARIN, 2012 [11] ................................................................. 33
3.5 Open water diagram for the full RANS computations of the Ka4-70 propeller in 19B duct, MARIN 2012 [11] ................................................................................................................... 33
3.6 Detachment of the tip-leakage vortex at the gap between the blade and the duct. The vortex is here visualized by isosurface of the Q-factor, and the colors refer to the contour levels of pressure coefficient. Adapted from Willemsen [12] ................................................................. 34
3.7 Contour plots of the vorticity showing flow separation at the outer duct surface and the recirculation region. $J=0.8$, MARIN, 2012 [11] ................................................................. 34
3.8 Open water diagram, with and without leading edge roughness for the Ka4-70 in 37 duct. MARIN, 2004 [13] 35
3.9 Open water diagram for the full RANS results of the Ka4-70 propeller in 37 duct. The CFD results and experimental from the Ka-propeller series are provided. Warstila for the CRS workgroup, 2013 [14]. ................................................................. 36
3.10 Chordwise pressure distribution on the duct at several angular location, together with computed mean values. Dataset from [14]. MS: model scale. ................................................................. 37
4.1 Paneling on the duct, the hub and the blades for Procal simulations of the Ka4-70 propeller in 19A duct. ................................................................. 40
4.2 Sketch of the 19A duct geometry and the iterative scheme for the modified trailing edge. . . 41
4.3 Convergence history for the radial trailing edge position. J=0.5, initial guesses at $\frac{T E}{L} = 0.020, 0.021$. 42
4.4 Geometries for the smallest and largest $y/L_{TE}$ tested as initial condition, and converged geometry for $J=0.5$. ................................................................. 42
4.5 Mean duct pressure distribution and axial force for the initial guesses and the converged solution, J=0.5. ................................................................. 43
4.6 Open water diagram and radial force for the initial guess at $\frac{T E}{L} = 0.020$ and the converged solution. ................................................................. 43
4.7 Open water diagram for the reference ($\frac{T E}{L} = 0.031$) and the iterative trailing edge geometry. . . 44
4.8 Section of the original 19A duct and the modified duct geometry for the reference Procal computations. ................................................................. 44
4.9 Comparison of the open water loading for the geometries with small and large hub. . . . . . . . 45
4.10 Open water diagrams comparing transpiration on the whole inner side (source model 3) and transpiration from blade TE to duct TE (source model 2). Diagrams are given for a set displacement thickness of $\delta_{TE}/L_{duct} = 0.02, 0.04$. ................................................................. 46
4.11 Effect on propeller thrust (left axis) and duct radial force (right axis) of variation of the displacement thickness $\delta_{TE}$ . ................................................................. 47
4.12 Radial force, Procal solution with $\delta_{TE}/L_{duct} = 0.005, 0.04, 0.08$ and ReFresco solution. For $\delta_{TE}/L_{duct} = 0.005$ and J=0.2 convergence problems were observed. ................................................................. 47
4.13 Relative difference in the loading with an increasing $\delta_{TE}$ taking the solution without transpiration as reference. Percentages give the relative difference in the radial force at the x-axis and the relative difference in thrust (both propeller and duct thrust) at y-axis. The duct radial force varies roughly 1.5 times the propeller thrust and 4.5 times the duct thrust. ................................................................. 47
4.14 Chordwise pressure distribution on the duct at blade angle of 0° and 45° with increasing value of displacement thickness at the trailing edge. Procal computations and full RANS results for J=0.5. ................................................................. 49
4.15 Chordwise pressure distribution on the duct at blade angle of 0° and 45° with increasing value of displacement thickness at the trailing edge. Procal computations and full RANS results for J=0.8. ................................................................. 49
4.16 Sketch of the regions where the pressure is mispredicted in the Procal solution. The red arrows give the vectors ($p_{RANS} - p_{BEM}$). For J=0.5 the misprediction in pressures balance each other out. For J=0.8 the duct thrust is overestimated. ................................................................. 49
4.17 Blade pressure contours for the Procal reference solution and the full RANS solution, J=0.5. . . 50
4.18 Location of the slices at constant $x/R$ for the duct angular pressure distribution. The duct extends from $x/R = 0.5$ to $x/R = 0.5$. The propeller tip section ($r/R \approx 0.99$) is shown in black. ................................................................. 51
4.19 Angular pressure distribution on the duct at J=0.5, Procal solution with $\delta_{TE}/L_{duct} = 0.04$, full RANS solution and experimental results corrected in phase. $x/R = 0.25$ (top left), $x/R = 0.075$ (top right), $x/R = -0.075$ (bottom left), $x/R = -0.3$ (bottom right). ................................................................. 52
5.1 (a)Percentage of volume of the BEM volume which is interpolated to the RANS grid with an increasing number of refinement steps in the VI interpolation method. (b) Relative difference in the duct loading with respect to the case with the largest number of refinement step investigated. ................................................................. 54
5.2 Convergence of the duct thrust and duct radial force with an increasing number of refinement steps in the VI scheme for interpolation of propeller loading. ................................................................. 55
5.3 Grid for the RANS part of the hybrid approach showing the surface mesh density and the body force distribution (the grid has 3.5M cells). ................................................................. 55
5.4 Grid refinement study for the RANS-BEM calculations of Ka4-70 propeller in 19A duct. ................................................................. 57
5.5 Mesh geometry and boundary condition for the RANS-BEM calculations of Ka4-70 propeller in 19A duct. ........................................... 57
5.6 Caption ............................................................................. 59
5.7 Comparison of BEM and RANS-BEM solution. Open water diagram for $D_{HUB}/D = 0.204$. ............................................. 59
5.8 Duct pressure distribution at $J=0.5$. Mean BEM Reference solution, mean BEM coupled solution and axisymmetric RANS-coupled distribution compared to a full RANS solution. ................ 60
5.9 Blade pressure distribution at $J=0.5$ at radii $r/R = 0.5, 0.9$. BEM reference solution, BEM coupled solution and full RANS distribution. ................................................................. 61
5.10 $J=0.5$. Streamlines and contour plots of the body force in X direction and the pressure coefficient CPN. ......................................................... 61
5.11 $J=0.8$. Streamlines and contour plots of the dimensionless vorticity $\omega$ and the axial velocity. Separation at the duct outer surface is observed at large advance ratios. ............ 62
6.1 Grid convergence study for the Ka4-70 propeller in 37 duct; the paneling of the duct surface is refined in axial direction. $J=0.5$. ................................................................. 64
6.2 Grid refinement study for the Ka4-70 propeller in 37 duct. The duct paneling is refined in angular direction. ................................................................. 65
6.3 Original and modified duct geometry with the position of the points used in the iterative scheme for the trailing edge geometry to match the pressure. ................................................. 66
6.4 Geometries for the initial $y/L_T$ and converged geometry for $J=0.5$. ................................................................. 66
6.5 Spanwise thrust distribution on the propeller blade and mean pressure distribution on the duct surface for the initial conditions and the converged solution at $J=0.5$. .................. 67
6.6 Open water diagram for the BEM computations with the iterative scheme for the trailing edge and a fixed value of trailing edge radial position. ................................................................. 67
6.7 View of the original duct 37 profile and the modified geometry for the Procal reference calculations. ................................................................. 67
6.8 Blade pressure contours for the Procal reference solution and the full RANS solution[14], $J=0.5$. ................................................................. 69
6.9 Chordwise pressure distribution on the duct for the BEM and full RANS[14] solutions at angular position of 0° and 40°. The oscillatory distribution in the RANS is likely due to a poor interpolation. ................................................................. 69
6.10 Section of the original and modified duct geometry and surface paneling. The unit outward-pointing normal vectors are plotted. For the paneling of the original 37 duct, the corner points are chosen according to the standard definition of the 37 duct section. For the modified geometry, the paneling results from the surface used in the Procal simulations. ............ 70
6.11 Three mean pressure distribution used in the 2D integration of the pressure over the duct section. ................................................................. 71
7.1 Grid refinement for the RANS-BEM computations of the Ka4-70 propeller in 37 duct. ................................................................. 74
7.2 Open water diagram and radial force for the Procal base computations and the RANS-BEM computations of the Ka4-70 propeller in 37 duct. ................................................................. 75
7.3 Open water induction at two planes upstream of the propeller at $x/R = 0.3$ and $x/R = 0.5$. (Left) Axial and radial velocity distribution. (Right) View of the propeller blade and the plane locations. Ka4-70 in 37 duct, $J=0.5$. ................................................................. 77
7.4 Duct pressure distribution at $J=0.2$ and $J=0.5$. Mean BEM Reference solution, mean BEM coupled solution, axisymmetric RANS-coupled distribution and full RANS solution. Ka4-70 propeller in 37 duct. ................................................................. 78
7.5 Blade pressure distribution at $J=0.5$ at radii $r/R = 0.5, 0.9$. BEM reference solution, BEM coupled solution and full RANS distribution. Ka4-70 propeller in 37 duct. ................................................................. 78
7.6 $J=0.5$. Streamlines, and contour plots of the body force in X direction and pressure coefficient CPN. ................................................................. 78
7.7 $J=0.8$. Streamlines and contour plots of the dimensionless vorticity $\omega$ and the axial velocity. Separation at the duct outer surface is observed at large advance ratios. ................................................................. 79
8.1 $J=0.5$ Ka4-70 in 19A duct. Tip leakage vortex detachment angle from full RANS computations and experimental observations. Experimental data refer to cavitation number $\sigma = 1.3$. ................................................................. 83
8.2 $J=0.5$ Ka4-70 in 19A duct. Tip leakage vortex detachment location from full RANS computations and experimental observations. Experimental data refer to cavitation number $\sigma = 1.3$. ................................................................. 83
8.3 Left: Contour plot of the vorticity magnitude on a cylindrical section at $r/R = 1$ for $J=0.5$. The red region of large vorticity is related to the detachment of the tip leakage vortex. A kink in observed in the trajectory of the vortex (emphasized in the figure).  

8.4 Planes at four angular location used to investigate the interaction of tip leakage vortex and tip vortex.  

8.5 Vortex image system with two real and two imaginary vortices, and resulting induced velocities.  

8.6 Angular component of the vorticity on the 4 planes shown in figure 8.4. The white streak visible in plane 1 is the section of the blade tip. Two regions of high vorticity are observed: region $a$ is attributed to the tip leakage vortex and region $b$ to the tip vortex. $J=0.5$.  

8.7 Planes at several angular positions that are used to study the flow in the gap between the propeller and the duct.  

8.8 Contour plot of the axial velocity at the gap between blade and propeller for the plane at $0^\circ$ blade angle. $J=0.5$.  

8.9 View of the mesh for the full RANS computations of the Ka4-70 propeller in 19B duct[11] and zoom-in on the detail of the grid distribution at the blade tip. The grid is dense close to the walls (duct wall and blade surface) but it is coarser in the region of the domain where the vortex is observed to develop.
List of Tables

3.1 Geometry for the experimental tests for the Ka4-70 propeller in 19A duct. 30
3.2 Geometry and settings for the full RANS computations of the Ka4-70 propeller in 19B duct. 32
3.3 Difference between experimental and numerical results, full RANS computations Ka4-70 propeller in 19B duct. [11]. 32
3.4 Geometry and settings for the full RANS computations of the Ka4-70 propeller in 37 duct. MARIN, 2004 [13]. 35
3.5 Geometry and settings for the full RANS computations of the Ka4-70 propeller in 37 duct. Wartsiila for the CRS working group, 2013 [14]. 36
3.6 Relative error between the full RANS calculation of Ka4-70 propeller in 37 duct and the experimental data from the Ka-propeller series. Computed from the dataset of the CFD computations carried out at Wartsiila for the CRS work group, [14]. 36
3.7 Angular locations of the available slices and weights attributed to compute the mean. 37
4.1 Outline of the chapter: description of the tasks carried out. 39
4.2 Geometry and wake model settings for the Procal computations of Ka4-70 in 19A duct. 40
4.3 Blade, duct and hub meshing with values for the spacings. 40
4.4 Radial position of the trailing edge obtained with the iterative scheme for the pressure difference. * not converged. 42
4.5 Open water data for the geometries with small and large hub diameter. 45
4.6 Settings for the transpiration velocity model. Key name for Procal control file, description and options used for computations. 46
5.1 Outline of the chapter with a list of the tasks carried out and the propeller geometry used. 53
5.2 CPU time on 32 cores 2.4GHz and interpolated volume with increasing number of recursive steps. 54
5.3 Detail of the RANS grid and the geometry used in the sensitivity to the maxRefLev parameter. 54
5.4 Grid details, duct loading and CPU time for the RANS-BEM computation with 20 coupling steps and three grids, with different boundary layer resolution at the hub. Computations are run on 32 cores, with 2.4GHz per core. 56
5.5 Grid refinement for the RANS-BEM computations of the Ka4-70 propeller in 19A duct. Computations are run on 32 cores (except the finest grid on *128 cores) with 2.4GHz per core; simulations with 1 RANS-BEM coupling step. ★ not considered in the fit, hence the uncertainty is not computed. 56
5.6 Geometry, topology and settings for the chosen grid for the RANS-BEM computations of the Ka4-70 propeller in 19A duct. ★ See reference[15] for the settings of the control file for ReFRESCO and [7] for the theoretical background. 58
5.7 Relative change wrt experimental. RANS-BEM, small hub $D_{hub}/D = 0.167$. (In the model tests $D_{hub}/D = 0.204$). 58
5.8 Relative change wrt experimental. RANS-BEM, large hub $D_{hub}/D = 0.204$. (In the model tests $D_{hub}/D = 0.204$). 58
5.9 Relative change wrt experimental. BEM $D_{hub}/D = 0.204$. (In the model tests $D_{hub}/D = 0.204$). 60
5.10 Relative change wrt experimental. RANS-BEM $D_{hub}/D = 0.204$. (In the model tests $D_{hub}/D = 0.204$). 60
6.1 Settings for the grid refinement study for the paneling on the duct in axial direction. 64
6.2 Numerical error and uncertainty for the grid refinement in axial direction. CPU time is between 25 and 40 minutes, without a clear dependence on grid refinement. ★ not in the fit. 64
6.3 Settings for the grid refinement study for the paneling on the duct in angular direction. 65
6.4 Numerical error and uncertainty for the grid refinement in angular direction. ★ not in the fit. 65
6.5 Geometry and wake model settings for the Procal computations of Ka4-70 in 37 duct. 65
6.6 Blade, duct and hub meshing with values for the spacings. 65
6.7 Radial position of the trailing edge obtained with the iterative scheme for the pressure differ-
ence. * not converged. ................................................................. 66
6.8 Relative change wrt experimental. BEM, duct 37, base trailing edge position. ....................... 68
6.9 Relative change wrt experimental. BEM, duct 37, iterative trailing edge position. ................. 68
6.10 Results for the 2D integration of the mean pressure over the original and the modified duct
geometry. The experimental results are provided for the duct thrust (not available for the radial
force. The values for the pressure integration and the relative difference with respect to the
experimental result are listed as well as the result from the BEM method Procal. ....................... 71

7.1 Grid refinement for the RANS-BEM computations of the Ka4-70 propeller in 37 duct. Computa-
tions are run on 32 cores (except the finest grid on *128 cores) with 2.4GHz per core; simulations
with 1 RANS-BEM coupling step. * Not in the fit, so the numerical uncertainty is not computed. 74
7.2 Geometry, topology and settings for the chosen grid for the RANS-BEM computations of the
Ka4-70 propeller in 37 duct. *See reference[15] for the settings of the control file for ReFRESCO
and [7] for the theoretical background. The complete control file is given in the Appendix. .... 74
7.3 Relative change wrt experimental. BEM Ka4-70 in 37 duct. ................................................. 76
7.4 Relative change wrt experimental. RANS-BEM Ka4-70 in 37 duct. ................................. 76

8.1 Predicted and experimental angles at which the tip leakage vortex detaches with respect to the
projected chord length at the blade tip. ................................................. 83
8.2 Location at which the tip leakage vortex detaches according to the front view of the CFD results
and experimental observations. ...................................................... 83
Glossary

$C_T$  Thrust load coefficient.
$C_{pn}$  Propeller pressure coefficient.
$F_S$  Safety factor.
$J$  Advance ratio.
$K_Q$  Torque coefficient.
$K_T$  Thrust coefficient.
$L_{duct}$  Duct length.
$Q$  Q-factor.
$R$  Propeller radius.
$U_\infty$  Free stream velocity.
$U_\phi$  Numerical uncertainty.
$Z$  Number of blades.
$\Phi$  Potential function.
$S$  Strain rate tensor.
$\omega$  Vorticity vector.
$\tau$  Stress tensor.
$n$  Unit vector normal to the body surface.
$\delta^*$  Boundary layer displacement thickness.
$\delta_{ij}$  Kronecker delta.
$\eta_0$  Open water efficiency.
$\mu$  Dipole strength.
$\mu$  Molecular dynamic viscosity.
$\mu_t$  Turbulent (or eddy) dynamic viscosity.
$\overline{u}$  Mean value of the random variable $u$.
$\phi$  Disturbance potential.
$\rho$  Fluid density.
$\sigma$  Source strength.
$c$  Chord length.
$h$  Gap height.
$h_i$  Grid size.
$n$  Propeller rotation rate.
$u'$  Fluctuation of random variable $u$.
$u_{G}$  Mean velocity in the propeller-duct gap.
BEM  Boundary Element Method.
CFD  Computational Fluid Dynamics.
DNS  Direct Numerical Simulation.
GCI  Grid Convergence Index.
IPKC  Iterative Pressure Kutta Condition.
LDV  Laser Doppler Velocimetry.
LE  Leading Edge.
LES  Large Eddy Simulation.
MARIN  Maritime Research Institute of the Netherlands.
PIV  Particle Imaging Velocimetry.
QUICK  Quadratic Upwind Interpolation for Convective Kinematics.
RANS  Reynolds Averaged Navier-Stokes.
ReFRESCO  Reliable Fast RANS equations solver for Ships and Construction Offshore.
SVI  Swept Volume Interpolation.
TE  Trailing Edge.
VI  Volume Intersection.
Introduction

This chapter gives an introduction to this study. A general overview is presented in section 1.1. The working principle of ducted propellers is explained in section 1.2, together with the description of the physical phenomena involved and the computational tools for the prediction of the performance. Then, section 1.3 gives the research objectives for this work and finally the report outline is presented in 1.4.

1.1. Motivation

The ducted propeller is a propulsion system which is constituted by a propeller mounted into an annular nozzle. The nozzle has an airfoil cross-section which is most often uniform around the duct. The propeller rotates inside the nozzle, which is supported by the the ship hull or by the propulsion unit case. The use of ducted propeller started in the early decades of nineteenth century when the Kort Propulsion Company pioneered the concept of fitting a nozzle to increase the propulsion efficiency in heavy loading condition [16]. From then onward, the use of ducted propeller has been widespread for ships operating in heavy loading such as tug boats. As an example, figure 1.1 shows a ducted propeller mounted on a single screw tug boat; the duct is here supported from the top, where it is clamped to the hull skeg.

In recent years, the ducted propulsion has also increased in popularity in the maritime and offshore industry where use is made of podded and azimuthal propulsors, frequently in ducted arrangement. The podded and azimuthal propulsors allow for a rotation of the propeller axis such that the manoeuvrability of the vessel is enhanced. The difference between the two lies in the position of engine and shaft drive: if the motor is located in the hull, the system is referred to as azimuthal propulsor while for the pods normally an electric motor is included in the pod house[17]. Both are commonly used for tugs or for propulsion (and positioning) of offshore floating structures.

From the hydrodynamic point of view, the force acting on the nozzle depends on the shape and the operating condition: both a positive axial force (contributing to the total thrust of the system) or a negative axial force can occur. Furthermore, because of the effect of the duct, the velocity at the propeller plane is different than the inflow velocity for a conventional propeller, so optimal designs will largely differ in the case of open propeller or when a nozzle is fitted. Also, an optimal duct geometry is significantly different if the performance is important only for forward sailing or if also astern sailing is relevant. In the former case, the airfoil cross section will have a thin trailing edge; in the latter case, the trailing edge is more rounded and more similar in curvature to the leading edge, such that similar performances are expected when sailing in opposite directions. Oosterveld[2] gives a good overview

Figure 1.1: Ducted propeller for an ATHS tug boat. Damen Shipyard, online source [1]
of the most commonly used duct sectional profiles.

According to the purpose and shape of the nozzle, two types of ducts are distinguished: accelerating and decelerating ducts, as sketched in figure 1.2. The accelerating duct produces an acceleration of the flow towards the propeller. It relates to a positive axial force on the duct which contributes to the total thrust. As concerns the decelerating ducts, they are fitted to improve the cavitation performances. The flow towards the blades is decelerated so the total pressure increases and a retardation of cavitation inception is achieved. This can be important for example for naval ships where a reduction of cavitation-induced radiated noise can become important for tactical reasons.

Generally, to predict the performance of the propulsor, one option is to carry out model tests in the water basin. This requires the facilities to manufacture the model and to tests it. Furthermore, the design of ducted propellers requires the analysis of the interaction between the two objects, because both the velocities induced by the duct affects the propeller inflow as well as the propeller action influences the flow around the airfoil-shaped nozzle. Hence, the experimental setup are more complicated and expensive than the open water tests for open propellers: even when tests are limited to the measurement of the loading (not considering pressure measurements nor PIV-LDV) the setup requires multiple force sensors on the nozzle supports, as shown in MARIN’s 2013 tests[10] for example.

Another option is the use of Computational Fluid Dynamics (CFD) tools. The governing equations are derived from the fundamental physical laws and they are solved numerically by a discretization of the domain. The numerical solution leads to large systems of equations that are solved with the aid of (super) computers. In this way, flow properties like pressure and velocity are solved and the forces acting on the system are computed. Furthermore, in the post-processing phase (i.e. after the solution is obtained) computational tools allow for a detailed analysis of the results and an easy flow visualization. The CFD tools are distinguished by the governing equations that are solved; tools based on potential flow theory are well established and used for design, while viscous flow calculations have greatly increased in popularity in view of the increased computational power available at reasonable expense. However, the numerical solutions are intrinsically limited by the modeling assumption for the governing equations that are solved, hence it becomes important to evaluate the applicability of such methods. The next section explains the working principle of ducted propellers and also gives information on the available computational tools. Chapter 2 provides a detailed theoretical background.

### 1.2. Background

The duct mounted around the propeller changes the flow velocity towards the propeller plane, as it is exerting a force on the fluid. The use of the Actuator Disk Theory (as in Kuiper[3] for instance) helps in understanding the working principles of the ducted propellers. The actuator disk is the simplest representation of the propeller action and it is developed to evaluate the induced velocities by the propeller-duct system as well as the ideal efficiency of the propulsor. The Actuator disk theory is based on the laws of conservation of mass,
1.2. Background

conservation of impulse and the Bernoulli law. The model assumes that the propeller is substituted by a disk where a jump $\Delta p$ in pressure is present. The duct as such is included in the model. It is assumed that there are no rotational effects and the disk exerts an axial force only. Viscous forces are neglected and the flow is steady such that the Bernoulli equation is valid along a streamline. Because rotational and viscous losses are discarded, the efficiency of the system calculated with actuator disk theory is the ideal efficiency, which is the highest efficiency possible. With reference to figure 1.3, the section $A_0$ is assumed far upstream, section $A_2$ is far downstream and section $A$ corresponds to the propeller location. The streamtube (shown in red) is the control volume considered for the application of the conservation of impulse. The upstream pressure $p_0$, the upstream velocity $V$, the diameter $D$ and the pressure jump $\Delta p$ are fixed. The velocities $v_1$ and $v_2$ are induced by the propeller+duct system, which, together with the uniform inflow, make the total velocity at sections $A$ and $A_2$. The propeller thrust is obtained through the Bernoulli equation and the duct thrust (which is an external force for the control volume considered) is obtained by application of the conservation of impulse (a complete derivation is given in the Appendix):

$$T_p = \frac{1}{2} \rho v_2 A (2V + v_2)$$

$$T_d = \frac{1}{2} \rho v_2 A (v_2 - 2v_1).$$

(1.1)

It is seen that the duct force relates to the induced velocities at the propeller plane $v_1$ and in the wake $v_2$. Commonly in the design of ducted propellers, the thrust ratio $\tau$ is considered which is defined as the ratio between propeller thrust $T_p$ and total thrust:

$$\tau = \frac{T_p}{T}.$$  

(1.2)

The induced velocities and the ideal efficiency of the propulsor are derived as functions of $\tau$. For accelerating ducts, the nozzle provides a positive thrust such that $\tau < 1$, while the decelerating ducts have $\tau > 1$ because the duct thrust is negative. The results obtained with the actuator disk theory are valid for open propellers as well, when $\tau = 1$. The velocity at the propeller disk $v_p = V + v_1$ are obtained as a function of the thrust ratio:

$$\frac{v_p}{V} = \frac{1}{2\tau} \left[ 1 + \sqrt{1 + \tau C_T} \right]$$

(1.3)

where the thrust load coefficient $C_T$ is:

$$C_T = \frac{T}{\frac{1}{2} \rho V^2 A} = \frac{T}{\frac{1}{2} \rho V^2 \frac{\pi}{4} D^2}.$$  

(1.4)

Furthermore, the ideal efficiency of the propeller is obtained when the kinetic energy lost $E$ is taken into account:

$$\eta_i = \frac{V \cdot T}{V \cdot T + E} = \frac{2}{1 + \sqrt{1 + \tau C_T}}.$$  

(1.5)
Equation 1.3 shows that the velocities at the propeller plane are larger when the thrust ratio is larger (given the same \( C_T \)), while the open water efficiency decreases with larger \( \tau \). This is the case for the accelerating ducts. Figures 1.4 provide the velocity and efficiency as function of \( C_T \) as in equations 1.3 and 1.5 respectively, with a parametric variation of \( \tau \). The curves are representative for accelerating, decelerating duct and open propeller arrangements.

On the one hand, the momentum theory gives a simple interpretation of the effect of the duct. On the other hand, the physical phenomena involved in the flow around the propeller and the nozzle are not completely modeled with a simplistic Actuator Disk. First of all, the relative motion of the propeller and the nozzle is left out in the previous analysis. Secondly, the effects of viscosity are large especially at the duct surfaces. At the inner surface of the duct the boundary layer can become thick such that the tip of the propeller rotates within the boundary layer region. Secondly, depending on the duct geometry, flow separation can occur. The flow separates at the trailing edge of the duct when the trailing edge is blunt, with a recirculation region more downstream. Also, flow separation is observed at the duct outer surface when the propeller is operating in light loading, when the inflow velocity has a large axial component. Finally, viscous effects are dominant in the flow at the gap between the blade and the nozzle.

The gap flow has a highly complex pattern. It is influenced by the geometry of the blade at the tip (the chord-length and the blade thickness) as well as by the loading condition, which relates to the magnitude of pressure difference between the pressure side and the suction side of the blade. Furthermore, the relative motion between the wall and the blade involves the fluid to "leap over" the blade tip from a region of higher pressure to a region of lower pressure, with strong velocity gradients. The resulting shear layer tends to roll-up and generates vortical structures: when the propeller has a large chord-length at the tip, a so-called tip leakage vortex develops. This vortex is characteristic of ducted propellers and it is not present for open propeller configurations. The mechanism that leads to the generation of the tip-leakage vortex differs from the generation of the usual tip vortex. The tip vortex occurs also for open propeller and it generates from the spanwise change in circulation at the propeller tip, with the vortex detachment at the blade tip trailing edge. For the tip-leakage vortex the combined effect of the pressure difference and the relative rotation makes the flow "leaking" from the blade pressure side to the suction side more upstream than the tip trailing edge, such that the vortex detachment occurs between the leading edge and the mid-chord positions. Figures 1.5 and 1.6 give two sketches of the tip leakage vortex detaching at the blade tip of a ducted propeller with large chordlength at the blade tip. This vortex is experienced to be strong and it influences the blade tip loading as well as the pressure on the inner surface of the duct in the wake. Furthermore, it can lead to severe cavitation. In literature, few experimental investigations are available for ducted propellers (2003, Oweis and Ceccio[18] for instance) with the aim to evaluate the effect of the tip leakage vortex on cavitation. Instead, when the goal is to investigate the detailed vortex dynamics (detachment and trajectory) and the effect of the gap size, the tests are carried out for fixed hydrofoils, for which Particle Imaging Velocimetry (PIV) or Laser Doppler Velocimetry (LDV) is easier and variations of the gap size are possible with the same setup (Dreyer[19] for example). A wider literature on the leakage vortex is available in the more general context of turbomachineries. Indeed, the leakage flow for ducted marine propellers is comparable to the leakage flow in the compressor stage of...
axial turbines. In both cases the leakage flow is governed by the relative rotation and the pressure difference at the blade, even though for the turbines also the blade cascade (i.e. the sequence of the blades) plays a role, Lakshminarayana[20]. Due to the importance for performance breakdown or over-heating, the leakage flow has been widely studied for turbomachineries, for which also detailed CFD studies (such as Large Eddy Simulation (LES), You[21]) were carried out. It is then expected that the analysis of such phenomena will be object of future research also for the marine applications.

The prediction of propeller performance relies on model tests or numerical studies with computational fluid dynamics. The available numerical tools differ in the governing equations which are solved and the degree of modeling involved:

- Potential flow packages are well established tools in the maritime industry and they are used daily in the design stage and for optimization studies. The potential flow methods assume that the flow is inviscid and irrotational. Extra models can be coupled or integrated to account for the viscous effect, but nonetheless the potential flow methods have strong limitations when viscous phenomena dominate. A special class of potential flow method is the Boundary Element Method (BEM). In this case only the boundaries of the domain are discretized for the numerical computations. For a three dimensional geometry, the boundaries are surfaces which are discretized with a set of panels, hence the name panel methods also used to refer to this approach. The use of BEM for open propellers in open water and in-behind condition is wide, but this is not the case for ducted propellers. Potential flow calculations on ducted propellers have been recently developed and a thorough validation (especially in view of the viscous effects involved with the nozzle) is under development.

- Secondly, viscous flow calculations are used. These methods retain the viscous term in the Navier-Stokes laws. They require a discretization of the entire 3D domain that is divided in cells on which the governing equations are applied and solved. In presence of turbulent flows the range of physical time and space scales involved is wide. A numerical solution would then require an extremely fine discretization, which is feasible for any but few applications (this is the so-called Direct Numerical Simulation (DNS) approach). Instead, the effect of turbulence is modeled, allowing for a feasible discretization level. According to the modeling of turbulence, different classes of CFD methods are developed. The LES methods have been under development in the last decades but are still limited for practical applications because of their expense. Instead, Reynolds Averaged Navier-Stokes (RANS) methods are often exploited. These are the most common type of viscous flow computations for marine propellers, which can also predict cavitation extent when cavitation models are implemented. RANS calculations for ducted propellers have been performed in wetted [11][14] or cavitating condition [22]. The RANS modeling allows for taking the viscous effects into account, but limitations still remain, such as the accuracy in the prediction of large separation regions[22].

At the Maritime Research Institute of the Netherlands (MARIN) both a boundary element method, namely PROCAL, and a RANS code, namely ReFRESCO, are developed. The BEM package has been widely validated for open propellers but only in recent years it was developed to include the duct surface[6]. In addition, a further method is developed which combines the boundary element package and the viscous solver. This is
referred to as RANS-BEM (or hybrid) approach. In the hybrid approach, the domain for the RANS computation includes the nozzle and the driving shaft, but the propeller is substituted with a body force distribution. The strength of the body force distribution is given by the boundary element method. Chapter 2 gives the details of this approach. The hybrid approach largely reduces the computational cost when compared to RANS-only simulations, and still the viscous effects at the duct surface are detected.

To sum up, the tools available at MARIN allow three possibilities for the numerical prediction of ducted propeller performance:

1. **BEM**: the inviscid, irrotational flow is solved for the duct and propeller system. The approach is computationally cheap but the results can be affected by the limitations of a potential flow analysis.

2. **RANS-BEM**: the viscous flow around the duct is solved when the effect of the propeller is substituted by a distribution of body forces. The method is currently developed for open water condition only.

3. **full RANS**: the viscous flow around the propeller-nozzle system is solved at once, taking into account the relative motion of the two objects. This approach is the most expensive but is expected to provide accurate results.

1.3. Objectives

The choice of one of the aforementioned approach is dictated by the required accuracy, the validity of the method for different geometries and the computational costs. Potential flow or hybrid schemes are attractive for design purpose or for the optimization process in view of the reduced costs. This study is concerned with the validation and development of the tools Procal and Procal-ReFRESCO for the analysis of ducted propellers. Two tests cases are considered which differ in the geometry of the duct. The same propeller is used, a Ka4-70 propeller, designed for operating in ducted arrangement, fitted in a 19A and a 37 nozzle[23] for the first and second test case. Both of them are accelerating ducts, but they largely differ in the trailing edge geometry, with nozzle 37 designed for good performances also when sailing astern. The duct geometries are given in the Appendix. These two tests cases allow a validation of the results because experimental tests and full RANS computations are available for both. The objective of this study are then set:

- Investigate the accuracy of the boundary element method Procal for predicting the ducted propeller performance for the two cases: Ka4-70 propeller in 19A duct and Ka4-70 propeller in 37 duct. The goal is to validate the results for propeller force, duct forces and efficiency against the experimental results and the available full RANS calculations. Moreover, a detailed analysis of pressure distributions allows for a thorough comparison of the potential flow results with the viscous computations, with the objective of understanding the limitations of using BEM for ducted propellers.

- Evaluate the improvements with the hybrid RANS-BEM approach in comparison to the potential flow calculations for the predicted forces and efficiency. Also, carry out a comparison of the hybrid approach with the more expensive full RANS computations, with the objective of investigating the effect of modeling the propeller with an axisymmetric body force distribution.

Moreover, the full RANS calculations are available for the first test case[11]. One further task is then set for this research:

- analysis of the full RANS calculations of the Ka4-70 propeller in 19A duct(2012) with focus on the predicted flow behavior in the gap between the propeller and the duct. The objective is to compare the full RANS results and the experimental observations as well as gain a further understanding of the structure of the tip leakage flow. The ultimate scope is to give guidelines on the modeling of the gap flow in a potential flow context.

To conclude, this research is well inserted in the context of the development of CFD tools for the analysis of ducted propellers, in which MARIN is involved through the PRODUCT\(^1\) project. The next section gives the outline of this report.

\(^1\)The PRODUCT project is an on-going project of the CRS (Cooperative Research Ship) workgroup, dealing with the development of Procal and hybrid methods for the prediction of the performances of ducted propellers. Information are available on the website: http://www.crships.org/web/show/.
Chapter 2 gives a description of the theoretical background. The governing equations are derived from the fundamental physical laws. The background for the boundary element method and the Reynolds Averaged Navier Stokes equation is explained, with a description of the computational tools available at MARIN. Chapter 3 introduces the experimental data and the full RANS computations which are used in this work for validation. The available data for the two tests cases are presented, with an explanation of how the data have been processed, where it was needed. The chapter also gives an overview of the geometry used in this study. Chapter 4 presents the results of the BEM and hybrid computations for the Ka4-70 propeller in 19A geometry. The focus is on the predicted loading and the pressure distributions on the blade and on the duct. In addition, the chapter explains a sensitivity study for the so-called transpiration velocity model implemented in Procal; the sensitivity study will allow to draw considerations on the coupling of Procal with the RANS code for the hybrid approach. Next, the results for the RANS-BEM calculations on the Ka4-70 propeller in 19A geometry are given in chapter 5. Seemingly, chapters 6 and 7 illustrates the BEM and RANS-BEM simulations respectively for the Ka4-70 in 37 duct. For this tests case, less validation data are available, but nonetheless the open water loads and the pressure distributions are analyzed. The analysis of the full RANS computations from 2012 is carried out in chapter 8 where the tip leakage vortex detachment location and downstream development are studied. The conclusions and the recommendations are provided in chapter 9. Finally, for the sake of readability, extensive derivations and some plots not provided in the previous chapters are included in the appendix.
2

Theoretical background and numerical tools

This chapter presents the theoretical background for the boundary element methods and the RANS-based methods as well as the implementation in the codes Procal and ReFRESCO. The first section introduces the governing equation, starting from the conservation laws of mass and momentum. From the conservation of mass, the Laplace equation is then derived, which is the basis for the boundary element methods, whereas the RANS equations are obtained by Reynolds-averaging the conservation of momentum. Sections 2.2 and 2.3 explain the detail of the in-house MARIN’s computer codes Procal and ReFRESCO, followed by the description of the coupling for the hybrid RANS-BEM in 2.4. The last sections 2.6 and 2.7 give respectively the theory for the grid refinement study and a review of the quantities used for the analysis of the performance of ducted propeller.

2.1. Governing equations

In this section the Navier-Stokes equations are introduced and a breakdown of the principles behind potential flow theory and RANS equations is presented.

2.1.1. Navier-Stokes equations

The Navier-Stokes equations provide the conservation of mass and momentum. A thorough derivation is not provided in this context, so reference is made to basic fluid dynamic books such as Kundu [24] or Katz[4]. The equations are provided in this chapter in differential form. The principle of conservation of mass, stating that mass is not created nor destroyed, leads to the continuity equation (2.1):

$$\frac{\partial \rho(x,t)}{\partial t} + \nabla \cdot (\rho(x,t) u(x,t)) = 0,$$

where \( \rho \) is the density, \( u \) the velocity vector and \( x \) the position vector of the fluid element. Using Einstein notation with repeated indices, equation (2.1) becomes:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0.$$

Working out the divergence operator in equation (2.1) and exploiting the definition of material derivative (\( \frac{D}{Dt} = \frac{\partial}{\partial t} + u_i \frac{\partial}{\partial x_i} \)), the equations is rewritten as

$$\frac{1}{\rho} \frac{D\rho}{Dt} + \frac{\partial u_i}{\partial x_i} = 0$$

which leads to the conservation of mass for a fluid with constant density, stating that the flow is divergence free:

$$\frac{\partial u_i}{\partial x_i} = \nabla \cdot u = div u = 0.$$
This is valid under the assumption of incompressibility, i.e. at small Mach number. Furthermore, it is valid when the Boussinesq approximation holds, according to which the relative density change in equation (2.3) are small compared to the velocity gradients in the divergence term, Kundu [24].

As concerns the conservation of momentum, the Newton second law relates the net force to the rate of change in (linear) momentum. Line, surface and body forces are acting on a fluid element. Line forces, such as surface tension, do not appear in the equations of motion that contains the body and the surface forces only. The surface forces are given in terms of stresses (i.e. force per contact area over the fluid element) by mean of the stress tensor \( \tau_{ij} \). The Newton second law leads to the equation of motion (2.5), commonly known as Cauchy’s equation of motion, here given for a velocity component \( u_i \):

\[
\rho \frac{Du_i}{Dt} = \rho g_i + \frac{\partial}{\partial x_j} (\tau_{ij}),
\]

(2.5)

where, at the left hand side, the rate of change of momentum is recognized, and at the right hand side there are the net body (\( \rho g_i \)) and surface forces (\( \partial \tau_{ij} / \partial x_i \)). To close the problem, the constitutive equation must be provided, that relates the strain rate tensor \( S_{ij} \), defined as

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

(2.6)
to the stress tensor \( \tau_{ij} \). For a Newtonian fluid (i.e. a fluid for which a linear stress-strain relation exists) this involves a fourth order tensor with 81 independent components, which can however be largely simplified considering that the stress tensor is symmetric and assuming an isotropic material so that the stress-strain relations is described by a single parameter \( \mu \), the dynamic viscosity. The constitutive equation is then given by:

\[
\tau_{ij} = -p \delta_{ij} + 2 \mu S_{ij}
\]

(2.7)

where \( p \) is the pressure, \( \mu \) the dynamic viscosity and \( \delta_{ij} \) the Kronecker delta.

Substitution of the constitutive relation in the Cauchy equation leads to the Navier-Stokes momentum equation, which for incompressible flow and for the component \( u_i \) is

\[
\rho \frac{Du_i}{Dt} = \rho g_i - \nabla p + \mu \nabla^2 u_i
\]

(2.8)

or, in vector notation,

\[
\rho \frac{D\mathbf{u}}{Dt} = \rho \mathbf{g} - \nabla \mathbf{p} + \mu \nabla^2 \mathbf{u}.
\]

(2.9)

The conservation equations (2.4) and (2.9) describe the motion of Newtonian, incompressible fluids.

### 2.1.2. Potential flow theory and the boundary element method

Assuming that the flow is irrotational, the continuity equation for incompressible fluid leads to the Laplace equation. When there is no vorticity (\( \omega = \nabla \times \mathbf{u} = 0 \)) it is possible to define the potential \( \Phi \) such that

\[
\mathbf{u} = \nabla \Phi.
\]

(2.10)

By substitution of (2.10) in the continuity equation for incompressible flow (2.4), the Laplace equation is obtained:

\[
\nabla \cdot \mathbf{u} = \nabla \cdot \nabla \Phi = \nabla^2 \Phi = 0,
\]

(2.11)

which is linear in the potential function \( \Phi \), meaning that complex flow patterns can be constructed from elementary solution, given the proper boundary conditions.

In addition, also without the assumption of irrotationality, the Navier Stokes momentum equations simplify to the Euler equations when assuming inviscid flow. Furthermore, rewriting the body force using the associated potential and working out the convective term (Kundu [24]), the Euler equation is re-written and the Bernoulli function \( B \) is recognized:

\[
\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j} \left( \frac{u_i^2}{2} + gz + \frac{p}{\rho} \right) = (\mathbf{u} \times \omega)_i
\]

\( B, \text{Bernoulli function} \)

(2.12)
The index $j$ is a repeated index over the velocity components, and the body forces are limited to gravity force for simplicity. At the right hand side is the cross product of the velocity and the vorticity $\omega$.

For a steady flow, considering that the cross product $u \times \omega$ is orthogonal to both $u$ and $\omega$, equation (2.12) results in the Bernoulli equation for rotational flow:

$$\frac{1}{2}u_j^2 + gz + \frac{p}{\rho} = \text{constant along streamlines & vortex lines}$$

(2.13)

Furthermore, if the flow is also irrotational (and steady), the Bernoulli function $B$ is constant everywhere in the field:

$$\frac{1}{2}u_j^2 + gz + \frac{p}{\rho} = \text{constant everywhere}$$

(2.14)

As concerns unsteady flows, to obtain a similar expression, the assumption of irrotational flow is needed to obtain the so-called ”unsteady Bernoulli” equation:

$$\frac{\partial \Phi}{\partial t} + \frac{1}{2} |\nabla \Phi|^2 + \frac{p}{\rho} + gz = F(t)$$

(2.15)

which is the expression used in potential flow theory for unsteady flows.

Because the Laplace equation is linear, arbitrary flows can be made up of elementary solutions. These elementary solutions can be considered as “building blocks” that are superimposed to generate complex flow patterns. The building blocks are defined by their associated potential functions, whose gradient gives by definition the velocity field. An example of elementary solution for a 2D case is the 2D uniform flow, whose potential with respect to a cartesian frame of reference is

$$\Phi(x, y) = Ux + Vy.$$  

(2.16)

Another, more interesting, example is the point source, that can be considered as a point from which flow is ejected. In a 2D case its potential is:

$$\Phi = \frac{m}{2\pi} \ln \sqrt{(x - x_0)^2 + (y - y_0)^2}.$$  

(2.17)

where $(x_0, y_0)$ is the position of the point source and $m$ the strength of the source. It is noticed that $\Phi$ is singular at the location of the source. This is the reason why such elementary solution is referred to as a singularity. Other singularities are the irrotational vortex and the doublet[24]. For these elementary solutions, when the distance gets to zero, the velocity becomes singular. A general solution method for the potential flow problem consists on distributing some selected singularities over the domain and finding the strength of the singularities such that the boundary conditions are satisfied. For open flows two boundary conditions are always present. At the body surface, the no-leakage condition must hold, which requires a zero normal velocity:

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

(2.18)

with $n$ the normal vector. In addition, the disturbance far from the body has to vanish:

$$\lim_{r \to \infty} (\nabla \Phi - \nu) = 0$$

(2.19)

with $r$ the position vector and $\nu$ the relative velocity between the body and the surrounding fluid, Katz,Plotkin [4].

Moreover, in presence of lifting bodies, a further condition is necessary, the Kutta condition. According to the Kutta-Joukowski theorem[4], the force generated by a lifting body (e.g. a wing profile) is related to the circulation $\Gamma$ induced in the flow. But, in the context of potential flow, the circulation remains constant (Kelvin theorem, Kundu [24] or Katz,Plotkin [4]). Therefore, when the body generates lift, a wake surface attached to the body is required, which extends downstream and carries the circulation produced. Such wake surface is

\[\text{For the sake of completeness, it is noted that for incompressible and irrotational flow the diffusion term in Navier-Stokes vanishes:}\]

$$\nabla^2 u = \nabla (\nabla \cdot u) - \nabla \times (\nabla \times u) = 0,$$

\[\text{hence the derivation of the Bernoulli from Euler equation is legitimized.}\]
force-free and is modeled by a distribution of singularities. The strength of the singularities in the attached wake sheet is not uniquely defined. Indeed, the solution for the lifting problem is not unique, since it is always possible to add circulation $\Gamma$ while the boundary conditions and the Laplace equation still hold, as shown in figure 2.1. Therefore, a further condition, the Kutta condition, is required to determine the amount of circulation (hence lift) produced.

The Kutta condition states that the flow at a sharp trailing edge leaves the edge smoothly and the velocity there is finite. So the solution is the one that provides just enough circulation to satisfy the Kutta condition. In order for the Kutta condition to be satisfied, the pressure at the trailing edge has to be continuous:

$$\Delta p_{T.E.} = 0. \quad (2.20)$$

This condition holds for a two-dimensional foil. In a 3D case the Kutta condition will apply for each cross section of the wing, hence over the whole trailing edge curve. The Laplace equation, boundary conditions (2.18) and (2.19) and the Kutta condition allow to describe the potential flow around lifting or non-lifting bodies. To determine the singularities, a general approach is based on Green's second identity, which leads to a relation between the potential at a point $P$ and the singularity distribution on the boundaries. A complete derivation is given in the Appendix. When $P$ is located on the surface of the body, the potential is:

$$\Phi(P) = -\frac{1}{2\pi} \int_{S_B} \left[ \sigma \left( \frac{1}{r} \right) - \mu \frac{\partial}{\partial n} \left( \frac{1}{r} \right) \right] dS + \frac{1}{2\pi} \int_{S_W} \left[ \mu \frac{\partial}{\partial n} \left( \frac{1}{r} \right) \right] dS + \Phi_\infty(P) \quad (2.21)$$

where $\Phi_\infty$ is the undisturbed potential, $\sigma$ and $\mu$ are the strengths of the so called source and doublet distribution. $S_B, S_W$ are the body and wake surfaces. When point $P$ is not located at the body surface, but in the fluid domain a factor of $1/4\pi$ instead of $1/2\pi$ precedes the integral signs. The doublet $\mu$ and the source $\sigma$ are related to the potential $\Phi$ by

$$-\mu = \Phi - \Phi_i \quad (2.22)$$
$$-\sigma = \frac{\partial \Phi}{\partial n} - \frac{\partial \Phi_i}{\partial n} \quad (2.23)$$

with $\Phi_i$ the internal potential that is normally set to zero.

The second integral of (2.21) is defined over the wake surface $S_W$ which is the thin surface attached to the body. Equation (2.21) is at the basis of the boundary element methods (BEM) and the solution for the potential problem consists on finding the singularity distribution over the boundaries such that the boundary conditions are satisfied. Nevertheless, the choice of singularities is not unique, so combination of sources/sinks, doublets or vortex can be used. As concerns the panel method Procal, more insight is given in section 2.2.
2.1.3. RANS equations

When the flow regime is turbulent, the velocity and the pressure fluctuate randomly, because of the turbulent eddies that have a range of length, time and energy scales. Figure 2.2 shows the measured signal of the axial velocity at the centerline of a fully developed turbulent jet. This is representative for a typical signal of the velocity in a turbulent flow. The signal is fluctuating and the mean value is shown as a solid line. Furthermore, figure 2.3 displays the dimensionless velocity profile in the jet as a function of the dimensionless cross-flow distance from the centerline. Such profile is obtained from via time average and dimensional analysis of the axial velocity. When dealing with turbulence flows, the statistical analysis of the fluctuating signals is necessary. The RANS equations are derived by time-averaging the Navier Stokes equations. The derivation follows here the same steps as in Pope[5].

For a stochastic turbulent flow variable $u$ (e.g. a velocity component), the time average is defined as:

$$\overline{U}(t) \equiv \frac{1}{T} \int_{t}^{t+T} u(\tau) d\tau,$$  \hspace{1cm} (2.24)

with $T$ the time length of the signal. Also, when $N$ samples of the flow can be obtained, the ensemble average is defined as:

$$\overline{U}^{N} \equiv \frac{1}{N} \sum_{n=1}^{N} u^{n}(t).$$ \hspace{1cm} (2.25)

Although time and ensemble average differ, when the flow is statistically steady (i.e. the probability distribution functions do not depend on time), the stochastic process is ergodic and it holds that

$$\lim_{T \to \infty} \overline{U}^{T} = \lim_{N \to \infty} \overline{U}^{N} = \overline{u}. \hspace{1cm} (2.26)$$

The length of the sample cannot be infinite, but $T$ has to be taken large enough to average out the turbulent fluctuations, so it must be much larger than the largest time scale of the turbulent macrostructures. The derivation of the RANS equations is based on the Reynolds decomposition, according to which a random variable is decomposed in the mean $\overline{u}$ and the fluctuation $u'$:

$$u = \overline{u} + u',$$ \hspace{1cm} (2.27)

The average operator has four properties: given two generic random variables $f$ and $g$, it is:

$$f + g = \overline{f} + \overline{g}$$ \hspace{1cm} (property1) \hspace{1cm} \begin{align*}
\overline{cf} &= c \overline{f} \\
\frac{\partial f}{\partial s} &= \frac{\partial \overline{f}}{\partial s} \\
\overline{fg} &= \overline{f} \overline{g}
\end{align*}$$ \hspace{1cm} (property2) \hspace{1cm} (property3) \hspace{1cm} \begin{align*}
\overline{f} + g &= \overline{f} + \overline{g} \\
\overline{cf} &= c \overline{f} \\
\frac{\partial f}{\partial s} &= \frac{\partial \overline{f}}{\partial s} \\
\overline{fg} &= \overline{f} \overline{g}
\end{align*}$$ \hspace{1cm} (property4)
Taking \( g = 1 \) in property (4) it also results that:

\[
\overline{f} = \overline{f} \tag{2.32}
\]

The time average does not satisfy property (4), but, if \( T \) is much larger than the time scale of the large eddies, the error is small, Pope\[5\]. Time-averaged values are then taken as mean values for the random flow variable such that the average operator correspond to the time average. The averaging operator is now applied to the continuity and momentum equation.

First of all, the continuity equation for incompressible flow (2.4) is averaged and worked out by use of the above properties and the definition of Reynolds decomposition:

\[
\frac{\partial \overline{u_i}}{\partial x_j} \overline{u_i} = \frac{\partial (\overline{u_i} + u_i')}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} = 0, \tag{2.33}
\]

Reynolds decomposition

which means that the mean velocity is divergence free.

Before averaging the momentum equations, the convective term in 2.8 is written in conservative form:

\[
u_j \frac{\partial u_i}{\partial x_j} = \frac{\partial}{\partial x_j} (u_j u_i) - u_j \frac{\partial u_i}{\partial x_j}, \tag{2.34}
\]

such that the left hand side of the averaged Navier-Stokes momentum equations is:

\[
\frac{\partial \overline{U_i}}{\partial t} \overline{u_i} + \frac{\partial}{\partial x_j} (\overline{u_i u_j}) = \overline{\mu u_i + u_j + u_i u_j}. \tag{2.35}
\]

Working out the argument of the advection derivative, it is:

\[
\overline{u_i u_j} = (\overline{u_i} + u_i') (\overline{u_j} + u_j') \overset{(1)(4)}{=} \overline{u_i u_j} + \overline{u_i u_j} + u_i' \overline{u_j} + u_j' \overline{u_i}. \tag{2.36}
\]

As concerns the right hand side of Navier Stokes equation, it only contains linear terms in \( u_i \), so the averaging is straightforward by use of properties (1) to (4). By substitution of (2.37) in (2.35) and re-arranging the terms in the averaged momentum equation, it is:

\[
\left( \frac{\partial \overline{u_i}}{\partial t} + \overline{u_j} \frac{\partial \overline{u_i}}{\partial x_j} \right) = -\frac{1}{\rho} \frac{\partial \overline{p}}{\partial x_i} + \nu \frac{\partial^2 \overline{u_i}}{\partial x_j^2} - \frac{\partial \overline{u_i u_j}}{\partial x_j}. \tag{2.38}
\]

This is the Reynolds averaged Navier-Stokes momentum equation for the \( u_i \) velocity component.

After some algebra for the right hand side, the previous expression is re-written as:

\[
\rho \left( \frac{\partial \overline{u_i}}{\partial t} + \overline{u_j} \frac{\partial \overline{u_i}}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left( \rho \overline{\delta_{ij}} \right) - \frac{\partial}{\partial x_j} \left( \rho \left( \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) + \rho \overline{u_i u_j} \right). \tag{2.39}
\]

Within brackets at the right side, the stress tensor appears together with an additional term \( \rho \overline{u_i u_j} \). This is the Reynolds shear stress and it is the stress due to the turbulent fluctuations. It shows up as a consequence of the averaging procedure. It is also noticeable that the Reynolds shear stress comes from averaging the convective term, even though it is put together with the diffusive term. This explains how the turbulent stress relates to the transport of mean momentum by turbulent fluctuations, induced by the eddies observed over a large range of length scales in turbulent flows.

In the RANS equations\(2.39\) there are 10 unknowns (3 velocities, the pressure and 6 Reynolds stresses) but 4 equations are given: the continuity equation and the 3 momentum equation, per each velocity component. Therefore, further equations for the Reynolds shear stress are needed to close the problem. A class of closure models for the Reynolds shear stress is based on similarity between molecular and turbulent stress. The Reynolds stress is expressed as follows:

\[
\rho \overline{u_i u_j} = \frac{2}{3} \rho k \delta_{ij} + \rho \left( \overline{u_i u_j} - \frac{2}{3} \rho k \delta_{ij} \right), \tag{2.40}
\]

isotropic part, turbulent pressure

deviatoric part, turbulent shear stress
where $k$ is the turbulent kinetic energy defined as:

$$k \equiv \frac{1}{2} \mathbf{u}' \cdot \mathbf{u}' = \frac{1}{2} \mathbf{u}'_i \mathbf{u}'_i$$  \hspace{1cm} (2.41)

The Boussinesq hypothesis, relates the deviatoric part of the Reynolds stress to the mean strain rate tensor $\overline{\mathbf{S}}_{ij}$:

$$\rho \left( \mathbf{u}'_i \mathbf{u}'_j - \frac{2}{3} \rho k \delta_{ij} \right) = -\rho \nu_t \left( \frac{\partial \overline{\mathbf{u}}}{\partial x_j} + \frac{\partial \overline{\mathbf{u}}}{\partial x_i} \right)$$  \hspace{1cm} (2.42)

$$= -2\rho \nu_t \overline{S}_{ij}. \hspace{1cm} (2.43)$$

$\nu_t$ is the turbulent kinematic viscosity (the turbulent dynamic viscosity $\mu_t$ can also be used equivalently for incompressible flows $\nu_t = \frac{\mu_t}{\rho}$). The order of magnitude of the turbulent viscosity relates to the product of the velocity and length scale of large eddies, $\nu_t \sim u_0 l_0$. Therefore, the closure of the problem is shifted from the resolution of the Reynolds shear stresses to the solution of the turbulent viscosity. To complete the set of equations, turbulence models are required to specify the value of $\mu_t$. Such models ranges from 0-equation models with constant $\mu_t$ to 2-equations models, where $\mu_t$ is a function of some turbulent quantities. Examples of well-known two equations models are the $k-\varepsilon$ or $k-\omega$ models which introduce additional equations for the turbulent kinetic energy $k$, turbulent dissipation $\varepsilon$ or dissipation rate $\omega$.

Detailed explanation of turbulence modeling is given by Wilcox[25] and those implemented in the RANS package ReFRESCO used in this work are presented in section 2.3. As a last remark, the Boussinesq approximation is valid when the production of turbulent kinetic energy by Reynolds stress is in local equilibrium with the dissipation $\varepsilon$. However, in close proximity to solid walls, where transport terms of turbulent kinetic energy become relevant, the validity of the Boussinesq hypothesis is questionable, Wilcox [25]. More advance models do not make use of the turbulent eddy approach, instead further equations for the Reynolds shear stress are defined, hence these models are commonly referred to as Reynolds stress models.

2.2. Procal

This section presents the panel method Procal developed at MARIN for prediction of propeller performance. The first part is dedicated to the general implementation, whereas the following sections focus on the wake alignment methods and treatment of lifting bodies with blunt trailing edge geometries, being those aspects relevant for the analysis of ducted propellers.

Procal is a low order panel method that implements the general approach as described in section 2.1.2, according to the general solution in equation (2.21). The complete theoretical background of Procal is found in [26]. Sources and dipoles are distributed over the propeller, hub and duct surfaces and dipoles are distributed in the thin wake surfaces. The condition for the disturbance potential $\phi$ to vanish at infinite distance is given as in equation (2.19) and the impermeability boundary condition follows from equation (2.18). Assuming that the internal potential equals zero, the impermeability condition leads to:

$$\nabla \phi \cdot \mathbf{n} = (\nabla \phi + \U_{\infty}) \cdot \mathbf{n} = 0 \Rightarrow \sigma = \nabla \phi \cdot \mathbf{n} = -\U_{\infty} \cdot \mathbf{n}, \hspace{1cm} (2.44)$$

so the source strength $\sigma$ of each panel is known once the normal vector to the blade surface and the free stream velocity $U_\infty$ is specified. Furthermore, by definition of doublet (equation (A.12)) for a zero internal potential it is:

$$\mu = -\phi. \hspace{1cm} (2.45)$$

therefore the dipole strength $\mu$ is the unknown and gives the value of the disturbance potential $\phi$.

In addition, the Kutta condition is needed to prescribe the value of the dipole at the wake. The Kutta condition relates the dipole strength at the wake with the difference of dipole strengths on the panels at lower and upper side of the trailing edge:

$$\mu_W = \mu_U - \mu_L = -(\phi_U - \phi_L) = -\Delta \phi_{T,E}, \hspace{1cm} (2.46)$$

which must hold over the whole trailing edge line. Nevertheless, the condition is based on a 2D approach and (2.46) might not suffice to ensure that the pressure is continuous at the trailing edge in a 3D case. Therefore, the dipole strength is found by the so-called Iterative Pressure Kutta Condition (IPKC). It requires the solution of the non linear equation

$$\Delta C_{p_{TE}}(\Delta \phi_{TE}) = 0, \hspace{1cm} (2.47)$$

with $\Delta C_{p_{TE}}$ the difference in pressure coefficient at the trailing edge. The iterative pressure Kutta condition is solved by mean of a Newton-Raphson scheme with the 2D Kutta condition as initial value [26].
2.2.1. Wake alignment

The strength of the doublet for the wake sheet is determined by the IPKC; however, the location and shape of the wake surface needs to be defined. The prediction of propeller performance has been proved to be largely influenced by the shape and location of the wake surface, for instance by Kinnas[27]. The issue of defining the wake location and shape is known as wake alignment. A good description of the theoretical background for the wake alignment is given by Katz[4].

According to the Kutta-Jouwkowski theorem the force acting on a body or surface immersed in a flow is proportional to the vector product of local velocity and the circulation: $\gamma$:

$$F = \rho u \times \gamma.$$  \hspace{1cm} (2.48)

The wake surface is required in the potential flow modeling, but the forces acting on the wake sheet must be zero. The wake sheet is force free when

$$F = 0 \Leftrightarrow u \times \gamma_W = 0 \Rightarrow \gamma_W \parallel u,$$  \hspace{1cm} (2.49)

The equation states that the wake surface must be aligned with the local velocity vector. The wake alignment procedure consists in adapting the position of the wake sheet such that condition (2.49) is satisfied.

More specifically for the BEM package Procal, the wake is modeled by a doublet distribution. Considering the relation

$$\gamma_W = -\nabla \mu_W$$  \hspace{1cm} (2.50)

proved in Katz[4], the condition for the alignment of the wake wake becomes:

$$u \times \nabla \mu_W = 0$$  \hspace{1cm} (2.51)

that must hold for each panel at the wake surface.

The wake alignment procedure requires an iterative procedure. An initial wake geometry is prescribed at the first step. The velocities are then computed and the new position of the wake surface is determined using the local velocity at the collocation points. The iterative wake alignment continues until the convergence criterion is met; this is usually taken as the root mean square of the distance of the wake points between two successive iterative steps.

More specifically for Procal, the alignment is based on a combination of so-called active update and passive update [28]. The active update occurs for the panels that are in the nearfield of the propeller and it is enforced by use of a cylindrical coordinate system $x, r, \theta$. The pitch of the wake sheet is updated at each iterative step to consider the local velocity:

$$P = 2\pi r_i^{(n)} \left( V_x(x_i^{(n)}, i, j, \theta_i^{(n)}), \frac{\partial V_x}{\partial \theta}(x_i^{(n)}, i, j, \theta_i^{(n)}) \right),$$  \hspace{1cm} (2.52)

with the superscript $n$ and the subscript $i$ referring to the alignment step and the index of the panel corner point respectively. The computed pitch of the wake sheet is used to update the positions of the panel points.

To ensure robustness of the code, the axial and radial positions are kept constant, while the angular position is updated. Therefore, the position of the panel corner points in the $(x, r, \theta)$ system is the following:

$$\begin{cases}
    x_i^{(n)} = x_i^{(n-1)} \\
    r_i^{(n)} = r_i^{(n-1)} \\
    \theta_i^{(n)} = \theta_i^{(n-1)} + 2\pi \frac{x_i^{(n-1)} - x_i^{(n-2)}}{x_i^{(n-1)} - x_i^{(n-2)}}
\end{cases} \quad i = 2, ... , N_{pw} \quad j = 1, ... , N_{rw}$$  \hspace{1cm} (2.53)

$N_{pw}$ is the number of points in the wake surface along the pitchwise direction and $N_{rw}$ is the number of points in radial direction. The index referring to the pitchwise direction starts from 2 because the points with constant $i=1$ correspond to the the trailing edge of the propeller, whose location is fixed for the alignment methods.

In addition, in order to avoid jumps in the geometry, the panels downstream of each active update are aligned according to a passive update scheme. The idea is that the active update computes the actual alignment, whereas the passive update passes the changes in wake geometries to the more downstream panels. The update of a downstream point $P$ is based on the displacement of the actively updated point $A$. A local
coordinate system \( s, t, n \) is used where \( x_s \) and \( x_t \) are the unit vectors tangential to the wake sheet at a certain location, \( x_n \) is the unit vector normal to it normal to it. In a local coordinate system, the displacement of the passively updated point is equal to the displacement of the last actively updated point. However, the difference between the local and global (\( x, y, z \)) coordinate system must be taken into account. A coordinate transformation from the propeller coordinate system to the local wake system is needed. Letting \( A \) an actively updated corner point and letting \( P \) a passively updated point downstream, the transformation relates the global displacement \( \delta x_P \) to the local displacements \( (\Delta s, \Delta t, \Delta n) \).

\[
\delta x_P = \Delta s x_P^s + \Delta t x_P^t + \Delta n x_P^n
\]  

and the displacements in the local reference \( (\Delta s, \Delta t, \Delta n) \) are given by the actively updated points. Figure 2.4 gives the wakes shape at convergence of the wake alignment scheme whereas figure 2.5 gives the flow diagram for the wake alignment procedure.

Moreover, when Procal is used for ducted propellers, an additional panel strip is inserted at the gap between the blade and the duct (see section 2.2.2). This surface is also paneled and a wake surface is attached in the same way as for the propeller blade. Therefore, the edge of the wake sheet lies on the duct inner surface. To avoid numerical problems, the panel distribution on the inside of the duct is matched with the wake surface, so that it is avoided that collocation points on the wake gets close to corner points on the duct[6].

### 2.2.2. Gap model

For the Procal computations of the ducted propeller performance, a panel sheet is inserted in the gap to avoid singular behavior. For the additional gap panels either the impermeability condition or a gap-model boundary condition is applied[6]. In the first case, the gap surface is treated as a solid boundary, no flow is allowed and the boundary condition for perturbation potential is that of equation (2.18). Differently, when the gap model is applied, a flow is permitted and the boundary condition is modified by introducing a transpiration velocity. The gap model is based on the two dimensional orifice equation and the gap losses are included by defining a discharge coefficient \( C_Q \):

\[
C_Q = \frac{Q_G}{h} \sqrt{\frac{\rho}{2\Delta p}}
\]

where \( Q_G \) is the volumetric flow rate through the gap, \( h \) the gap height and \( \Delta p \) is the pressure difference across the gap. Expressing the flow rate in terms of the gap velocity \( u_G \) it results:

\[
u_G = C_Q \sqrt{\frac{2\Delta p}{\rho}} = \left| U_\infty \right| C_Q \sqrt{\Delta C_p}
\]
so the boundary condition is modified to take into account the transpiration velocity:

$$\frac{\partial \phi}{\partial n} = -\mathbf{U}_\infty \cdot \mathbf{n} + |\mathbf{U}_\infty| \sqrt{\Delta C_p} \mathbf{n} \cdot \mathbf{n}_c,$$

(2.57)

with $\mathbf{n}_c$ the unit vector normal to the mean camberline at the gap strip at the same chordwise position. Moreover, because the pressure depends on the velocity field, the gap model requires an iterative scheme to have equation (2.57) satisfied. This is done by taking as a first estimate the solution with closed gap and computing the pressure from the potential by a finite difference scheme. Finally, it is important to bear in mind that the gap model depends on the prescribed value of the discharge coefficient, so the solution is affected by this empirical parameter.

### 2.2.3. Duct trailing edge

The sectional profiles of the nozzle often present a blunt trailing edge. Both test cases investigated in this study have (ducts 19A and 37) have a blunt trailing edge. For the Procal computations, the duct trailing edge is modified to a sharp geometry to enforce the wake alignment procedure. BEM simulations with a blunt trailing edge would be possible with a prescribed wake model. However, in that case there would be a large pressure recovery at the blunt trailing edge because flow separation is not modeled in the potential flow analysis. This would lead to a misprediction of duct thrust. The geometry of the duct must be modified to provide a sharp trailing edge. As will be described in chapter 4, the change in geometry requires a modification of the duct inner and outer surface profiles close to the trailing as well as setting both axial and radial position of the trailing edge. It was proved in former researches[12],[8] that the effect of the radial position of the trailing edge is the quantity that mostly affects the loading on the duct. At the current stage of development, Procal does not provide any model to find an appropriate modified geometry. The user has to choose for an appropriate change in trailing edge position, on the basis of sensitivity studies or analysis of flow behavior from viscous CFD computations[12]. In this research, an alternative model is provided, which involves a pressure-equality criterion. This model is based on the work of Kinnas [27] and its application for the two test cases is given in chapter 4 for duct 19A and chapter 6 for the duct 37.

### 2.3. ReFRESCO

In this section the package ReFRESCO is described, which is used in this work for the hybrid RANS-BEM calculations. The Reliable Fast RANS equations solver for Ships and Construction Offshore (ReFRESCO) is the U-RANS finite-volume code developed at MARIN. In this section, the theoretical background of ReFRESCO is
treated, with focus on the turbulence modeling and the discretization methods; a thorough explanation of theory behind ReFRESCO is given by Vaz and Hoekstra [7]

The governing equations are solved in ReFRESCO in a segregated manner, with a Poisson equation for the pressure correction[7]. Figure 2.6 gives the working diagram for the numerical solution. The outer iteration levels couple the equations for the momentum, pressure correction and turbulence modeling. The inner loops are required to solve the non-linearity, by a Picard-type linearization[29]. Because ReFRESCO is also capable of computing the multi-phase flow, additional transport equations for the volume fraction for different species are considered. This step is also presented in the flow diagram for completeness, but the multi-phase flow analysis is not carried out in this study because cavitation is not considered. Finally, the solution diagram shows the procedure for an unsteady computation, for which an initial solution is required and the time step has to be incremented within the time loop. However, it is important to bear in mind that the computations in this work will be with steady flow, since the case of open water condition is considered.

2.3.1. Turbulence models

As explained in section 2.1.3 the RANS equations requires a closure model. The Boussinesq hypothesis (2.43) relates the Reynolds stress to the turbulent viscosity \( \nu_t \). To close the problem, ReFRESCO provides one equation or two equations models: the one equation Spalart-Allmaras model, \( k-\epsilon \) models and \( k-\omega \) models.

As concerns the \( k-\epsilon \) models, variants of the original Launder & Sharma (1974) are implemented in ReFRESCO. In these models the eddy viscosity relates to turbulent kinetic energy \( k \) and dissipation rate \( \omega \) by

\[
\nu_t = C_{\mu}\frac{k^2}{\epsilon}. \tag{2.58}
\]

The standard \( k-\epsilon \) model provides two equations for the turbulent kinetic energy \( k \) and dissipation rate \( \epsilon \):

\[
\frac{\partial k}{\partial t} + \nabla \cdot \tau_{ij} = \frac{\partial \tau_{ij}}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \tag{2.59}
\]

\[
\frac{\partial \epsilon}{\partial t} + \nabla \cdot \tau_{ij} = C_{\epsilon 1} \frac{\epsilon}{k} \frac{\partial \tau_{ij}}{\partial x_j} - C_{\epsilon 2} \frac{\epsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_{\epsilon}} \right) \frac{\partial \epsilon}{\partial x_j} \right] \tag{2.60}
\]

where \( P \) indicates the production of turbulent kinetic energy by Reynolds shear stress acting on the mean flow, and \( D \) is the viscous dissipation. \( C_{\mu}, C_{\epsilon 1}, C_{\epsilon 2}, C_{\nu}, \sigma_k, \sigma_{\epsilon} \) are constants of the model. Details of such models for ReFRESCO are given by Hoekstra[7].

Another popular two equations model is the \( k-\omega \). It provides equation for the turbulent kinetic energy \( k \) and for the specific dissipation rate \( \omega \). This is defined as:

\[
\omega = \frac{\epsilon}{k}. \tag{2.61}
\]

which represents a characteristic frequency of turbulence. The turbulent viscosity is obtained as

\[
\nu_t = C_{\omega} \frac{k}{\omega}. \tag{2.62}
\]

Again, there are different versions of this model implemented in ReFRESCO. The original model was introduced by Wilcox [25] and is based on the two following equations:

\[
\frac{\partial k}{\partial t} + \nabla \cdot \tau_{ij} = \frac{\partial \tau_{ij}}{\partial x_j} - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[ \left( \nu + \sigma^* \nu_{\epsilon} \right) \frac{\partial k}{\partial x_j} \right] \tag{2.63}
\]

\[
\frac{\partial \omega}{\partial t} + \nabla \cdot \tau_{ij} = \alpha \frac{\omega}{k} \frac{\partial \tau_{ij}}{\partial x_j} - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \nu + \sigma^* \nu_{\epsilon} \right) \frac{\partial \omega}{\partial x_j} \right] \tag{2.64}
\]

with \( C_{\omega}, \alpha, \beta^*, \sigma, \sigma^* \) the constants of the model.

In addition, the \( k-\omega \) SST model is implemented in ReFRESCO. It is a combination of the \( k-\omega \) and the \( k-\epsilon \) models. It makes use of auxiliary functions to adapt the model to the region of fluid considered: close to wall,
2. Theoretical background and numerical tools

Initial solution \( t=0 \)

Increment \( t \)

Solve momentum equations

Solve pressure correction

Solve turbulence model equations

Solve additional transport equation

Converged?

\( t = \text{end?} \)

Post processing

Figure 2.6: Working diagram of ReFRESCO, adapted from [7].
it behaves like a standard $k-\omega$ model, whereas away from wall it behaves like a $k-\varepsilon$ model. The $k-\omega$ SST was introduced by Menter(1994). The turbulent viscosity is given by:

$$\mu_t = \frac{\rho k\omega}{\max(1, \Omega F_2/(\alpha_t \omega))}$$  \hspace{1cm} (2.65)

where $\Omega$ here is the flow vorticity to avoid confusion with the dissipation rate $\omega$. $F_2$ is an auxiliary function which is defined by

$$F_2 = \tanh \left\{ \max \left( 2 \sqrt{\frac{K}{0.09 d \omega}}, \frac{500 \mu}{\rho d^2 \omega} \right)^2 \right\}$$  \hspace{1cm} (2.66)

with $d$ being the wall distance. The transport equation for the kinetic energy is:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho U_j k - (\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right) = \tau_{ij} S_{ij} - \beta^* \omega k$$  \hspace{1cm} (2.67)

while the transport equation for the dissipation rate includes another auxiliary function $F_1$:

$$\frac{\partial \rho \omega}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho U_j \omega - (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right) = \gamma \rho \Omega^2 - \beta^2 \rho \omega^2 + 2(1 - F_1) \frac{\rho \sigma_{\omega^2} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}}{\omega}$$  \hspace{1cm} (2.68)

The auxiliary function and the coefficients of the model are given by:

$$F_1 = \tanh \left\{ \min \left( \max \left( 2 \sqrt{\frac{K}{0.09 d \omega}}, \frac{500 \mu}{\rho d^2 \omega} \right)^2 \right) \right\}^{1/4}$$  \hspace{1cm} (2.69)

$$CD_{kw} = \max \left( \frac{2 \rho \sigma_{\omega^2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 1 \times 10^{-20} \right)$$  \hspace{1cm} (2.70)

$$a_1 = 0.31, \quad \beta^* = 0.09, \quad \kappa = 0.41.$$  \hspace{1cm} (2.71)

Still the coefficients $\beta, \gamma, \sigma_k, \sigma_\omega$ have not been defined yet. These are obtained with a transition between the original $k-\omega$ (denoted by subscript 1) and the $k-\varepsilon$ models (subscript 2). Letting $\phi$ be one of the four coefficients, it results that

$$\phi = F_1 \phi_1 + (1 - F_1) \phi_2, \quad \phi = [\beta, \gamma, \sigma_k, \sigma_\omega]$$  \hspace{1cm} (2.72)

where specifically the coefficient for the two formulations are given by:

$$\sigma_{k1} = 0.85, \quad \sigma_{\omega1} = 0.50, \quad \beta_1 = 0.075, \quad \gamma_1 = \beta_1 / \beta^* - \sigma_{\omega1} K^2 / \sqrt{\beta^*} = 0.553,$$  \hspace{1cm} (2.73)

$$\sigma_{k2} = 1.00, \quad \sigma_{\omega2} = 0.856, \quad \beta_2 = 0.0828, \quad \gamma_2 = \beta_2 / \beta^* - \sigma_{\omega2} K^2 / \sqrt{\beta^*} = 0.440.$$  \hspace{1cm} (2.74)

The $k-\omega$ SST model is applied in this work for the RANS part pf the RANS-BEM hybrid approach. To conclude, a fully detailed review of the turbulence models implemented in ReFRESCO is made by Vaz and Hoekstra[7].

2.3.2. Discretization

ReFRESCO implements a finite volume method, with a cell-centered based and co-located grid approach[7]. In the finite volume method, the discretization of volume and surface integrals is required. Given a generic scalar variable $\phi$, the volume integral is computed directly from the value $\phi_C$ at the cell center:

$$\int_V \phi dV \approx \phi_C \Delta V.$$  \hspace{1cm} (2.75)

To compute the integrals over the surface of a 3D cell, the values of $\phi$ at the face and the surface areas are used:

$$\int_S \phi dS \approx \sum_{i=1}^{S_f} \phi_{f_i} S_{f_i},$$  \hspace{1cm} (2.76)

with $\phi_{f_i}$ the values at the faces. As concerns the spatial discretization of the gradients, from the Gauss divergence theorem the following relation holds\(^2\):

$$\int_V \nabla \phi dV = \int_S \phi n dS.$$  \hspace{1cm} (2.77)

\(^2\text{This can be proved easily by applying the divergence theorem to a vector field } \phi \hat{c} \text{ (with } \hat{c} \text{ constant unit vector)}\)
2. Theoretical background and numerical tools

Figure 2.7: Iterative scheme for the RANS-BEM coupling. Adapted from [8]

which, in the discrete case, provides the values for the gradients at the cell center:

$$\nabla \phi \approx \frac{1}{\Delta V} \sum_{i=1}^{N_f} \phi_{f_i} S_{f_i}. \quad (2.78)$$

For the computation of face centered values $\phi_{f_i}$, a simple interpolation between center values is made when faces are orthogonal to the center-to-center line, otherwise the eccentricity is taken into account. In the latter case an iterative scheme is necessary because the value at the face center is then dependent on the gradient at the face center and viceversa[7]. The discretization of the pressure and the body force terms follows from (2.75) while the discretization of the diffusive terms makes use of diffusion term is obtained from an equation of the form of (2.78). The convective term requires a special treatment for the face quantities. The convective term is given by

$$F_C = \int_S \phi (U \cdot n) dS, \quad (2.79)$$

that is discretized as

$$F_C \approx \sum_{i=1}^{N_f} (U_{f_i} \cdot S_{f_i}) \phi_{f_i} = \sum_{i=1}^{N_f} q_{f_i}^\phi, \quad (2.80)$$

where $q_{f_i}^\phi$ is the convective flux of $\phi$ through the face of the cell.

For the convective term the value of $\phi$ at the face is computed with different schemes. A hybrid scheme is available, that mixes the default central difference scheme with a first-order upwind scheme by mean of blend functions; alternatively, the Quadratic Upwind Interpolation for Convective Kinematics (QUICK) scheme is also available.

As concerns the time discretization, it is not presented in this section, rather reference is made to the work of Hoekstra and Vaz[7]. Briefly, the time derivative (for unsteady computations) are computed by using an implicit backward approximation schemes, with the options for a first-order backward Euler scheme or second-order backward difference.

2.4. Procal-ReFRESCO coupling

As pointed out in the introduction, large viscous effects occur at the duct, with flow separation at duct inner or outer surfaces and re-circulation at the trailing edge. At MARIN, a hybrid approach has been developed, that couples the potential flow analysis by Procal with RANS simulations by ReFRESCO.

Figure 2.7 shows a diagram of the iterative process. In Procal both the duct and the propeller are present. From the BEM a volume distribution of body forces induced by the propeller action is obtained. The pressure difference between the blade sides is projected to the camber surface to get a force distribution. The force is
2.4. Procal-ReFRESCO coupling

The carthesian mesh is representative for the BEM mesh the red polygon for a RANS cell.

Figure 2.8: Graphical illustration of the recursive method for volume intersection in two dimension. Adapted from [9].

then distributed over the so-called panel volume [9] which is the volume generated by connecting the corresponding panels of two successive blade positions. The volumetric force distribution over the whole swept volume of the propeller is interpolated to the RANS grid, with one of the interpolation methods described in section 2.4.1. Then, a RANS computation of the duct and hub plus body force takes place. This gives a value of the duct force which is compared with the duct force from Procal; if the difference is below the tolerance, the procedure is converged, otherwise a new Procal computation is run. For the successive Procal computation the effective camberline of the duct is changed by mean of a transpiration velocity model as explained in section 2.4.2 to match the value of the duct force coming from the RANS. Then, a new propeller force will result, which is interpolated to the body force in the next iteration step.

The procedure does not require evaluation of the effective wake, as it is in the case of the hull-propeller system, because the duct geometry is physically present in Procal. Also, even though the distribution of body forces is axisymmetric, ReFRESCO does not provide an axisymmetric formulation (at the moment) and therefore the RANS simulations must consider a 3D domain. Finally, either the radial force or the total force on the duct can be chosen as convergence criterion, [8]. In chapter 5 a sensitivity study for the transpiration velocity model in Procal will allow to draw conclusions on which option is preferable.

2.4.1. Body force interpolation

To interpolate the force field from Procal to ReFRESCO, different methods were developed to pass the volume force distribution from the panel volumes to the RANS cells:

- Nearest cell. A search algorithm finds the closest panel volume center for each RANS cell center, and the corresponding value of force-per-volume is assigned. The nearest cell was used for instance in [30] but was lately abandoned to prefer more sophisticated schemes.

- Swept volume interpolation. The value at each RANS cell center is assigned by trilinear interpolation among the neighboring panel volume center points. The method is more effective than the nearest cell method, but it suffers of an important drawback: when the BEM panel distribution is fine (such as at the leading edge) and a pressure peak is located at one of this panels, the contribution of the pressure peak will be spread over the large RANS cell which lies in the swept volume of this panel. Therefore, to avoid pressure peaks to be overestimated, a smoothing procedure is needed.

- Volume intersection. This method is based on an intersection algorithm for arbitrary volumes. The force per volume in the RANS cell is computed as the weighted average of the body force at the panel volumes that overlap with the RANS cell. The weighting factors are the amounts of overlapping. To determine the overlapping a recursive algorithm is implemented. Figure 2.8 illustrates the working principle of the algorithm for a 2D case. In the figure the cartesian mesh is assumed to be the BEM panel distribution, and the red polygon is a RANS cell. For each panel it is determined whether the panel is entirely located inside the RANS cell. If so, it will fully contribute. If not, the panel is subdivided in subpanels and for each subdivision it is again checked if it belongs entirely to the RANS cell. Proceeding in such a recursive way, the amount of overlapping is calculated. The number of recursive step is set by the user. This method increases the computational time but provides significant improvements in the computation, as shown in [9] for the hull-propeller system.
2.4.2. Transpiration velocity model
In the RANS-BEM coupling the duct force is matched between the potential flow and viscous solutions. The duct loading is changed in Procal by mean of a transpiration velocity model. The impermeability boundary condition (2.18) is modified to include a velocity $V_N$ that results in a change in the effective camberline of the duct profile, hence the loading distribution varies [8].

The transpiration velocity is given by

$$V_N = \frac{\partial}{\partial s}(U_e \delta^*), \quad (2.81)$$

where $\delta^*$ is the displacement thickness, $s$ the dimensionless curvilinear coordinate along the surface and $U_e$ the velocity at the edge of the boundary layer, assumed to be the free stream velocity (i.e. the velocity of the ship). The boundary layer displacement thickness is assumed to grow with $s$ up to the value at the duct trailing edge:

$$\delta^* = \delta_{TE}^* s^3. \quad (2.82)$$

Substitution of (2.82) in (2.81) leads to the final expression for the transpiration velocity:

$$V_N = 3U_e \delta_{TE}^* s^2. \quad (2.83)$$

The transpiration velocity is then used to determine the modified source strength:

$$\sigma = -U_\infty \cdot n + V_N. \quad (2.84)$$

Within Procal there are four options for the location where the transpiration model is applied: the outer surface of the nozzle, the whole inner surface of the nozzle, the inner surface from the blade trailing edge to the nozzle trailing edge and the inner surface from the nozzle leading edge to the blade trailing edge. Figure 2.9 gives a schematic illustration of the transpiration velocity model and the options available.

2.5. Current capabilities of Procal and the hybrid Procal-ReFRESCO methods
In this section few aspects are described about the current stage of development of Procal, ReFRESCO and the coupling Procal-ReFRESCO for the analysis of ducted propellers:

- At the current stage of development, only open water computations of ducted propellers are possible with Procal. The unsteadiness due to a ship wake cannot be taken into account yet.

- In the Procal-ReFRESCO coupling the ReFRESCO computation requires a 3D mesh because the package is not able to perform axisymmetric calculations, even though the geometry (shaft and nozzle) and the body force distribution are axisymmetric;
The wake alignment method developed for ducted propellers, with adaptation of wake and duct panels, is not very robust, with convergence issues arising especially for low advance ratios.

In view of the first consideration, only open water calculations are carried out in this work. As concerns the lack of an axisymmetric formulation for ReFRESCO, it results in larger computational time than it would be for the axisymmetric case, as well as the need to generate a 3D mesh.

2.6. Grid refinement theory

A grid refinement study is carried out in chapters 5 and 7 for the two tests cases tackled in this work. The algorithm for the computation of the error, uncertainty and convergence index follows the procedure described by Eça and Hoekstra[31].

The error of the numerical solution is the discrepancy from the exact solution while the uncertainty $U_\phi$ is an interval that contains the exact solution within a 95% range. The procedure consists of two parts: the calculation of the discretization error $\epsilon_\phi$ for a certain quantity $\phi$ is

$$\epsilon_\phi \approx \delta_{RE} = \phi_i - \phi_0 = a h_i^p$$

where $\phi_i$ is the value of the investigated quantity as computed with the grid $i$, $\phi_0$ is the estimate of the exact solution, $a$ a constant, $p$ the observed order of convergence and $h_i$ the cell size. Equation (2.85) has three unknown and it must hold for all the $n_g$ grids considered in the grid refinement; it is then solved with a least-square approach to obtain $a, p, \phi_0$. Equation (2.85) is a good error estimator only if the expansion of the error can be expressed with a single power $p$, i.e. if all the grids lies in the asymptotic range. This is hardly ever the case for practical applications. Therefore, the approach by Eça and Hoekstra[31] introduces two additional conditions:

- if the computed $p > 2$, two more power fits are solved (again in a least square sense):

$$\delta_1 = a h_i \quad \delta_2 = a h_i^2$$

- if the computed $p < 0.5$ or impossible to establish, three more power fits are computed:

$$\delta_1 = a h_i \quad \delta_2 = a h_i^2 \quad \delta_{12} = a_1 h_i + a_2 h_i^2$$

Formulations 2.86 and 2.87 allows to take into account practical application when the grid dataset has a poor quality, such as in presence of non-monotonic convergence. The error is taken from the fit that exhibits the smaller standard deviation $\sigma$.

Once the error is computed, the calculation of the uncertainty follows. The uncertainty $U_\phi$ is the interval that contains the exact solution with a 95% accuracy:

$$\phi_i - U_\phi \leq \phi_{exact} \leq \phi_i - U_\phi$$

The uncertainty is obtained from the value of the error, the value for the safety factor $F_S$ and an assessment of the quality of the fit. First of all, a so-called range parameter is computed:

$$\Delta_\phi = \frac{(\phi_i)_{max} - (\phi_i)_{min}}{n_g - 1}$$

After which the safety factor is set as:

$$F_S = 1.25 \quad \text{if} \quad 0.5 \leq p < 2.1 \quad \text{and} \quad \sigma < \Delta_{phi}$$

$$F_S = 3 \quad \text{otherwise}$$

The uncertainty is finally given by:

$$U_\phi(\phi_i) = F_S \epsilon_\phi(\phi_i) + \sigma + |\phi_i - \phi_{fit}| \quad \text{if} \quad \sigma < \Delta_\phi$$

$$U_\phi(\phi_i) = 3 \frac{\sigma}{\Delta_\phi} \epsilon_\phi(\phi_i) + \sigma + |\phi_i - \phi_{fit}| \quad \text{if} \quad \sigma \geq \Delta_\phi$$

The procedure assumes that the discretization error is dominant over the round-off error and the iterative error[32].
The uncertainty will differ with the grid (because the error is different), even though is common practice to refer to the uncertainty for the finest of the $n_g$ grids considered. Moreover, the error estimation assumes that the grids are described by a single parameter $h_i$. This implicitly assumes that the grids are self-similar, which means that the grid properties (deviation from orthogonality, skewness) are independent of the grid refinement\[32\]. The lack of grid similarity would lead to poor or misleading estimation of the error and uncertainty. Finally, to define the typical cell size, the relative step size is used. It gives the measure of the coarsening of a grid compared to the finest mesh available. Given that $n_i$ as the number of cells. The relative step size is defined as:

\[
\frac{h_i}{h_1} = \frac{n_1 - 1}{n_i - 1} \quad \text{for 1D grids}
\]
\[
\bar{h}_i = \sqrt{\frac{n_i}{n_1}} \quad \text{for 2D grids}
\]
\[
\bar{h}_i = \sqrt[3]{\frac{n_i}{n_1}} \quad \text{for 3D grids}
\]

The grid refinement studies result in the plots presented in chapters 5 and 7 where the solutions and the fit are given as function of the relative step size. In addition, the same procedure in used in chapter 5 to evaluate the effect of the refinement steps in the Volume Intersection method. In that case the refinement is treated as a refinement of a 1D mesh.

2.7. Analysis of propeller performance

This section provides some definition for the analysis of the performance of (ducted) propellers. The quantity presented will be used in the following chapters. The performance of the propeller are commonly analyzed using dimensionless parameters. The quantities that are used to define the reference values are the propeller rotation rate $n$, i.e. the rotations per second, the propeller diameter $D$ and the advance velocity $V_A$, which equals the speed of the ship when the propeller is in-behind condition or the uniform inflow for the open water calculations. The advance ratio $J$ expresses how much the propulsor advances in one revolution \[3\]:

\[
J = \frac{V_A}{nD}.
\]

Furthermore, the thrust $T$ and the torque $Q$ are made dimensionless by use of a reference length and a reference velocity. The reference length is the diameter while the reference velocity is the product of diameter and rotations rate, $V_{ref} = nD$. The thrust coefficient $K_T$ and the torque coefficient $K_Q$ are then obtained:

\[
K_T = \frac{T}{\rho V_{ref}^2 L_{ref}^2} = \frac{T}{\rho n^2 D^4}.
\]
\[
K_Q = \frac{Q}{\rho V_{ref}^2 L_{ref}^3} = \frac{Q}{\rho n^2 D^5}.
\]

The thrust coefficient can refer to the total thrust, the thrust of the propeller only, or the thrust of the duct only. In the three cases the thrust coefficient is defined as $K_{TT}$, $K_{TP}$ or $K_{TD}$.

Secondly, the open water efficiency $\eta_0$ is defined as the ratio of the effective power to the delivered power:

\[
\eta_0 = \frac{P_E}{P_D} = \frac{T V}{2\pi n Q} = \frac{J K_T}{2\pi K_Q}.
\]

Also, although in this research cavitating flow computations are not performed, it is useful to recall the definition of cavitation number:

\[
\sigma_n = \frac{p_{ref} - p_v}{\frac{1}{2}\rho V_{ref}^2} = \frac{p_{atm} + \rho g z_{shaft} - p_v}{\frac{1}{2}\rho n^2 D^2},
\]

with $z_{shaft}$ the shaft immersion and $p_v$ the vapor pressure at the given temperature. The subscript $n$ is used for the cavitation number when the hydrostatic contribution is also included.
Cavitation occurs when the pressure locally falls below vapor pressure. This condition can be expressed in terms of dimensionless quantities when considering the propeller pressure coefficient $C_{pn}$:

$$C_{pn} = \frac{p - p_{ref}}{\frac{1}{2} \rho V_{ref}^2}.$$  \hspace{1cm} (2.102)

Letting $p = p_s$ leads to the condition for the minimum pressure coefficient at the location of incipient cavitation:

$$\sigma_n = -C_p(min).$$  \hspace{1cm} (2.103)

Finally, two more definitions are provided which are used to analyze the viscous flow simulations. First of all, the so-called $Q$-factor, Jeong[33]. It is used to visualize the vortices and is defined as:

$$Q = \frac{1}{2} \left( \| \Omega \|^2 - \| S \|^2 \right)$$  \hspace{1cm} (2.104)

where $\Omega$ and $S$ are the vorticity and strain rate tensor respectively:

$$\Omega_{i,j} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)$$  \hspace{1cm} (2.105)

$$S_{i,j} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$  \hspace{1cm} (2.106)

and $\| \star \|$ is the trace of the tensors. The Q-factor is used in chapter 8 for the visualization of the tip leakage vortex from the full RANS calculations.

Secondly, for the contour plots of the vorticity produced in this work, the dimensionless vorticity magnitude is used which is equal to:

$$\omega' = \frac{\omega}{U_{ref} L_{ref}}.$$  \hspace{1cm} (2.107)

The reference velocity and length do not necessarily coincide with the reference length and velocity for the loading coefficients. When using the dimensional vorticity to study the flow around the nozzle, the reference velocity is taken as the advance velocity (i.e. uniform inflow), $U_{ref} = V_A$ and the reference length is taken as the duct length, $L_{ref} = L_{duct}$. Finally, the definition of dimensionless vorticity apply in the same way to the single components of the vorticity.

The next chapter (before presenting the results) is dedicated to the description of the experimental data and the full RANS computations used for validating the BEM and the hybrid methods.
In this chapter the experimental data and the results for the full RANS calculations are presented. These dataset are used in this work for validation purpose. Sections 3.2 and 3.3 give the results for the experiments and CFD on the Ka4-70 propeller in duct 19A, while sections 3.4 and 3.5 refer to the tests and computations with duct 37 respectively. Open water diagrams for the two test cases are presented.

3.1. Introduction

In order to validate the boundary element method and the hybrid RANS-BEM calculations, use is made of experimental data and full RANS calculations. This chapter is intended to show the data used for validation. It is out of the scope of the present work to provide a thorough analysis of these results. However, the CFD and the open water tests are compared to give a measure of the accuracy that the full RANS approach currently provides, which is useful for comparison with the boundary element and hybrid computations performed in this project.

Tests and CFD calculations were carried out for both geometries considered. CFD computations for the Ka4-70 in 19B duct were executed at MARIN in 2012,[11] whereas the most recent open water tests were run in 2013,[10] (including PIV, pressure measurements and cavitation observations). As concerns the Ka4-70 propeller in 37 duct, CFD calculations are available within the framework of the CRS (Cooperative Research Ship) workgroup and executed by Warstila, [14]. As for the experimental results, only open water tests are available for this geometry, from model tests run at MARIN in 2004 [13].

Furthermore, as regards the CFD computations for the 37-duct, the whole digital dataset is not available for further post-processing and therefore a limited validation with respect to pressure distribution and flow detail is possible. Differently, for the 19A-duct case, the CFD data are accessible and therefore a more complete validation is accomplished.

Finally, the geometry used in the experiments and CFD is not consistent for what concerns the hub diameter. Hub diameters of \( D_{\text{HUB}} / D = 0.167 \), \( D_{\text{HUB}} / D = 0.204 \) and \( D_{\text{HUB}} / D = 0.199 \) were used in the past for the tests and CFD. In this chapter the general data for each case are given in tabular form while the detailed geometry file is included in the Appendix. In view of this inconsistency in the propeller geometry used in the previous tests and CFD calculations, the effect of the hub diameter is also studied in this work, such that BEM and RANS-BEM computations are carried out for two hub diameter \( D_{\text{HUB}} / D = 0.167, 0.204 \), as reported in chapters 4 and 5.

3.2. Experimental data for the Ka4-70 propeller in 19A duct

Open water tests for the Ka4-70 propeller in 19A duct were run at MARIN in 2013[10]. These include PIV, pressure measurements at the duct surfaces and cavitation observations. Table 3.1 gives the main characteristics of the model while figure 3.1 shows the setup of the tests. The drive shaft extends downstream, with the thrust-torque sensors fitted in the hub; the nozzle is supported by three arms where force sensors are
mounted to measure the duct thrust. For the pressure measurements, a brass nozzle is used, fitted with pressure sensors whereas for the cavitation observations a transparent perspex nozzle is mounted. Furthermore, for the cavitation observations leading edge roughness is applied on the blade to reduce the scale effects on cavitation inception. The leading edge roughness results in a larger propeller torque and smaller propeller thrust. Therefore, there occurs a maximum difference in the open water efficiency up to 8% at $J \approx 0.55$. Because the computations carried out in this work are at model scale, and since there is no mention of laminar flow separation at the blade surfaces, it is decided to use the loading measured without leading edge roughness for validation. Figure 3.2 shows the open water diagram for two rotation rates ($n=900,1200\text{rpm}$). There is a small difference between the two, mainly in the open water efficiency $\eta_0$. For consistency with the CFD calculations, the values for $n=900\text{rpm}$ are used. The design condition for this propeller is between $J=0.5$ and $J=0.6$ where the efficiency is maximum.

As concerns the pressure measurements, 15 pressure sensors are mounted on the duct surface (5 on the outer surface, 10 on the inner surface). From these sensors, signals of the angular pressure distribution on the duct are obtained for several chordwise locations. A blade-to-blade variation is observed in the pressure measurements, which is most likely caused by a relative motion between the the propeller and the nozzle[10]; therefore, for validation the blade-rate ensemble average of the signals is taken such that the blade-to-blade variation is averaged out. Due to the few numbers of sensors in chordwise direction on the duct, it is not possible to extract the chordwise duct pressure distribution.

<table>
<thead>
<tr>
<th>Propeller model</th>
<th>Ka4-70</th>
</tr>
</thead>
<tbody>
<tr>
<td>Duct model</td>
<td>19A</td>
</tr>
<tr>
<td>Number of blades</td>
<td>$Z$</td>
</tr>
<tr>
<td>Diameter</td>
<td>$D$</td>
</tr>
<tr>
<td>Pitch ratio at 0.7R</td>
<td>$P_{0.7}/D$</td>
</tr>
<tr>
<td>Hub diameter ratio</td>
<td>$D_{HUB}/D$</td>
</tr>
<tr>
<td>Duct length</td>
<td>$L_{duct}$</td>
</tr>
<tr>
<td>Gap width</td>
<td>$h$</td>
</tr>
</tbody>
</table>

Table 3.1: Geometry for the experimental tests for the Ka4-70 propeller in 19A duct.

Figure 3.1: Front view of the setup for the open water tests for the Ka4-70 propeller in 19A duct. The shaft extends downstream and the orange tape minimizes the reflections of laser light during PIV measurements [10].
3.2. Experimental data for the Ka4-70 propeller in 19A duct

Figure 3.2: Open water diagram, measured at 900rpm and 1200rpm for the Ka4-70 in 19A duct. Adapted from [10].

Figure 3.3: Blade-to-blade variation in the cavitation pattern for the tests carried out at MARIN in 2013 [10]. Blades three and four are shown; the nozzle is transparent. Operating condition here is \( J=0.5 \) and \( \sigma=0.3 \). Differences are seen in the extent of sheet and bubble cavitation. For this work, it is relevant that there is a difference in the chordwise position at which the vortex begins to cavitate at the blade tip.
3.2.1. Cavitation observations

Cavitation observations are carried out for two advance ratio ($J=0.5, 0.2$) and four cavitation numbers ($\sigma=1.3, 1.5, 1.7, 1.9$), obtained by changing the ambient pressure in MARIN’s Depressurized Wave Basin. High-speed videos are recorded with two cameras located at the front and at the side on the propulsor. The cavitation observations are used in this work for comparing the prediction of the CFD computations with the experimental data for what concerns the behavior of the tip leakage vortex, particularly the detachment of the vortex and the downstream development (chapter 8). Therefore, for this analysis it is assumed that looking at cavitation is a valid criterion to detect where the vortex is located. This is not true in general, because the vortex core does not necessarily cavitates. However, the tip leakage vortex is expected to be strong enough that cavitation is occurring already at the detachment from the blade tip. In addition, it is decided to use the observations for the low cavitation number $\sigma=1.3$, when the flow is more prone to cavitating. Nevertheless, it is important to bear in mind that considerations on the appearance of vortical structure in the experiments are bounded to the occurrence of cavitation.

Moreover, a blade-to-blade variation is observed in the cavitation pattern. Figure 3.3 shows the cavitation pattern at $J=0.5$ and $\sigma=0.3$ for blade three and four. It is noted that the bubble cavitation pattern differ significantly between the two blades: for blade 3 bubble cavities are seen at radii larger than 0.7R while for blade 4 already at radii 0.5R. Furthermore, there is a vast region of cavitation at the propeller tip. This is related to a sheet cavity developing at the gap between blade and nozzle as well as the tip leakage vortex cavitation. What is relevant for the present work is the chordwise location (with reference to the chordlength at the propeller tip section) at which the cavitation is first occurring: for blade 3 cavitation appears at smaller chordwise position than for blade 4. As a result, to take into account the blade-to-blade variation, large uncertainties in the visual comparison of experiments and CFD (chapter 8) must be considered.

3.3. Full RANS computations of the Ka4-70 propeller in 19B duct

Full RANS calculations of the Ka4-70 propeller in 19B duct were carried out at MARIN in 2012 [11]. The 19B duct slightly differs from the 19A at the trailing edge, but the geometry is modified to simplify the meshing procedure such that the actual geometry used in the calculations is entirely equal to the duct 19A; a hub diameter of 0.167 times the propeller diameter is used. Table 3.2 presents the geometrical and numerical settings for the computations and figure 3.4 shows the grid distribution on the blade, duct and hub surfaces together with a view of the mesh in the domain. The RANS code ReFRESCO is used and the equations are solved in the rotating (non-inertial) reference frame, such that steady calculations are possible. As concerns the turbulence models, the $k-\omega$ SST model is used (described in chapter 2) and the grid has approximately 32M cells. The open water diagram is shown in figure 3.5 where a very good agreement is observed for the duct thrust and the open water diagram, even though both propeller thrust and torque are consistently underpredicted through the whole open water range. It is out of the scope of this work to investigate the reasons for this underpredictions, rather the accuracy of the full RANS approach is looked at. Table 3.3 lists the relative difference with respect to the experimental data. There is an underprediction from 3% to 7% in $K_{T_{prop}}$ and up to 6% for $K_{Q}$. The duct thrust also shows an underprediction up to 4%. As for the open water efficiency, there is a good agreement with an average of 2% difference with the experiments. Finally, large relative differences

<table>
<thead>
<tr>
<th>Propeller model</th>
<th>Ka4-70</th>
</tr>
</thead>
<tbody>
<tr>
<td>Duct model</td>
<td>19B</td>
</tr>
<tr>
<td>Number of blades</td>
<td>$Z$</td>
</tr>
<tr>
<td>Diameter</td>
<td>$D$</td>
</tr>
<tr>
<td>Pitch ratio at 0.7R</td>
<td>$P_{0.7}/D$</td>
</tr>
<tr>
<td>Hub diameter ratio</td>
<td>$D_{hub}/D$</td>
</tr>
<tr>
<td>Duct length</td>
<td>$l_{d_{dct}}$</td>
</tr>
<tr>
<td>Gap width</td>
<td>$h$</td>
</tr>
</tbody>
</table>

| Rotation rate | 900rpm |
| Turbulence model | $k-\omega$ SST |
| Momentum discretization scheme | QUICK |
| Turbulence discretization scheme | Upwind |
| Mesh size      | 31788 591 |

Table 3.2: Geometry and settings for the full RANS computations of the Ka4-70 propeller in 19B duct.

<table>
<thead>
<tr>
<th>$J$</th>
<th>$\Delta K_{T_{prop}}$</th>
<th>$\Delta K_{Q}$</th>
<th>$\Delta K_{T_{dct}}$</th>
<th>$\Delta \eta_{0}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.10</td>
<td>-2.63%</td>
<td>-2.45%</td>
<td>-1.59%</td>
<td>0.27%</td>
</tr>
<tr>
<td>0.20</td>
<td>-2.86%</td>
<td>-2.40%</td>
<td>-2.11%</td>
<td>0.65%</td>
</tr>
<tr>
<td>0.30</td>
<td>-3.52%</td>
<td>-2.47%</td>
<td>-3.15%</td>
<td>1.27%</td>
</tr>
<tr>
<td>0.40</td>
<td>-4.49%</td>
<td>-2.80%</td>
<td>-4.37%</td>
<td>1.99%</td>
</tr>
<tr>
<td>0.50</td>
<td>-5.62%</td>
<td>-3.28%</td>
<td>-3.42%</td>
<td>2.16%</td>
</tr>
<tr>
<td>0.60</td>
<td>-7.17%</td>
<td>-4.07%</td>
<td>-11.54%</td>
<td>-1.01%</td>
</tr>
<tr>
<td>0.70</td>
<td>-7.10%</td>
<td>-3.44%</td>
<td>13.86%</td>
<td>-2.16%</td>
</tr>
<tr>
<td>0.80</td>
<td>-11.84%</td>
<td>-5.99%</td>
<td>-5.71%</td>
<td>1.19%</td>
</tr>
</tbody>
</table>

Table 3.3: Difference between experimental and numerical results, full RANS computations Ka4-70 propeller in 19B duct. [11].
3.3. Full RANS computations of the Ka4-70 propeller in 19B duct

Figure 3.4: Grid distribution on the surface and on the domain for the full RANS calculations of the Ka-4-70 propeller in 19A duct. MARIN, 2012[11].

Figure 3.5: Open water diagram for the full RANS computations of the Ka4-70 propeller in 19B duct, MARIN 2012[11].
(> 10%) occurs whenever the quantities compared are very small (such as for $K_{T\text{prop}}$ at $J=0.8$ and $K_{T\text{duct}}$ at $J=0.6,0.7$). These values are not representative for a good comparison because in such cases the absolute difference is on the same order of the computed/measured values.

In addition, the entire dataset for these computations was made available such that a further post-processing of the results is possible to obtain detailed information for validation of the BEM and RANS-BEM calculations. Pressure distributions at the blade and at the duct are extracted and contour plots of pressure, velocity and vorticity are produced for validation. Furthermore, an important integral quantity computed is the radial force acting on the duct, which is obtained by integration of the pressure and the shear stress on the duct. Values for the radial force acting on the duct are valuable to make considerations on the duct loading, hence the velocities induced. For completeness, it is observed that the shear stress contribution in the radial force is between 2% and 3% while for the duct thrust it is between 3% and 4% depending on the advance ratio. Therefore, it is concluded that the contribution of the shear forces is small for the duct loading, hence a comparison of the pressure distribution on the duct with the potential flow calculations (were shear is not present) is representative for the integral forces. Finally, to compare the full RANS computations with the axisymmetric RANS-BEM, the mean pressure distribution on the duct is calculated by averaging 90 chordwise distribution corresponding to angular positions from 1 to 90 degrees (which corresponds to averaging over the entire circumference for this open water case with a four-bladed propeller).

3.3.1. Tip leakage vortex and flow separation at the duct surface

The full RANS results are used in this work also to carry out an analysis of the tip leakage vortex structure and the flow pattern in the gap. Figure 3.6 gives a visualization of the tip leakage vortex obtained from the CFD results. The vortex is visualized through the isosurface of the Q-factor (section 2.7) and it is here colored by the pressure coefficient level. The vortex detaches at about one third of the chord length from the blade leading edge, at the suction side of the blade. In the figure some streaks are also visible at the tip, which are due to the boundary layer developing at the duct wall (the nozzle is not shown here) where the Q-factor can become equal to the value used to visualize the vortex for the isosurface.

Moreover, the CFD calculations show that flow separation occurs at the duct outer surface for $J>0.6$. Figure 3.7 shows the contour of the vorticity magnitude at a plane at $0^\circ$. There is a region of large vorticity at the duct nose where separation occurs. The flow recirculation at more downstream locations gives rise to two regions of large vorticity where the flow is circulating. Even though it is not in the scope of this work to assess the capabilities of a RANS solver in predicting flow separation correctly, it is clear that the unsteadiness in flow separation are left out of this steady flow computation. Furthermore, what is more relevant for this project, is the fact that the separation pattern in the full RANS 3D case depends on the angular position. Differently, for the RANS-BEM hybrid approach, the case is axisymmetric and therefore the angular differences in the separating flow are lost. This is found to influence the results and likely be one cause for some mispredictions in the results (chapter 7).
3.4. Experimental data for the Ka4-70 propeller in 37 duct

For the duct 37 less validation material is available than for the 19A duct. Tests for the Ka4-70 propeller in 37 duct were run in 2004 at MARIN[13] during a series of tests with different propellers and duct. Only the open water tests are carried out, therefore there is no experimental measurement of pressure on the duct nor cavitation observations. Table 3.4 gives the geometrical details for the tests while figure 3.8 provides the open water diagram. Tests were run with and without leading edge roughness at the nozzle. With nozzle roughness the nozzle thrust is slightly smaller such that the open water efficiency is also smaller, but the maximum difference in $\eta_0$ is only 2%. For consistency with the experimental data for the duct 19A, it is decided to use the results without leading edge roughness.

<table>
<thead>
<tr>
<th>Propeller model</th>
<th>Ka4-70</th>
</tr>
</thead>
<tbody>
<tr>
<td>Duct model</td>
<td>37</td>
</tr>
<tr>
<td>Number of blades</td>
<td>$Z$</td>
</tr>
<tr>
<td>Diameter</td>
<td>$D$</td>
</tr>
<tr>
<td>Pitch ratio at 0.7R</td>
<td>$P_{0.7}/D$</td>
</tr>
<tr>
<td>Hub diameter ratio</td>
<td>$D_{HUB}/D$</td>
</tr>
<tr>
<td>Duct length</td>
<td>$L_{duct}$</td>
</tr>
<tr>
<td>Gap width</td>
<td>$h$</td>
</tr>
</tbody>
</table>

Table 3.4: Geometry for the experimental tests for the Ka4-70 propeller in 37 duct. MARIN,2004 [13]

3.5. Full RANS computations of the Ka4-70 propeller in 37 duct

As concerns the full RANS for the Ka4-70 propeller in 37 duct, calculations were carried out for the CRS (Cooperative Research Ship) PRODUCT workgroup by Warstila using the CFD package STAR-CCM+. The available
data for such computations are limited to the final report[14] and the dataset uploaded on the CRS website. Table 3.5 shows details of the geometry and the settings for the calculations. The mesh (with approximately 6 million cells) is much coarser than the one used for the 19A duct. The same turbulence model (\( k - \omega \text{SST} \)) is used. The open water diagram in figure 3.9 provides the CFD and experimental results while table 3.6 lists the relative difference. The duct thrust is underpredicted at mid advance ratios with a discrepancy of 4% while the propeller thrust is constantly overpredicted from +1% to 10% at large Js. These two mispredictions balance out so the total thrust is in very good agreement (<2%). However, the propeller torque is underpredicted with a 5% difference. This results in a visible over prediction of open water efficiency (from +1% to even +50%). As observed in section 3.3, there occur very large (but not meaningful) relative errors when the loading gets very small at larger advance ratios. The data used for validation also includes detailed pressure distribution on the blade and on the duct for two loading condition, J=0.2 and J=0.5. Unfortunately, values for the radial force acting on the duct (which relates to the duct induced velocities) are not provided and cannot be computed from the available data.

![Open water diagram for the full RANS results of the Ka4-70 propeller in 37 duct. The CFD results and experimental from the Ka-propeller series are provided. Warstila for the CRS working group, 2013 [14].](image.png)

<table>
<thead>
<tr>
<th>Propeller model</th>
<th>Ka4-70</th>
</tr>
</thead>
<tbody>
<tr>
<td>Duct model</td>
<td>37</td>
</tr>
<tr>
<td>Number of blades</td>
<td>Z = 4</td>
</tr>
<tr>
<td>Diameter</td>
<td>D</td>
</tr>
<tr>
<td>Pitch ratio at 0.7R</td>
<td>( P_{0.7}/D )</td>
</tr>
<tr>
<td>Hub diameter ratio</td>
<td>( D_{HUB}/D )</td>
</tr>
<tr>
<td>Duct length</td>
<td>( L_{duct} )</td>
</tr>
<tr>
<td>Rotation rate</td>
<td>900rpm</td>
</tr>
<tr>
<td>CFD package</td>
<td>STAR-CCM+</td>
</tr>
<tr>
<td>Mesh size</td>
<td>Approx 6M cells</td>
</tr>
</tbody>
</table>

Table 3.5: Geometry and settings for the full RANS computations of the Ka4-70 propeller in 37 duct. Wartsila for the CRS working group, 2013 [14].

<table>
<thead>
<tr>
<th>J</th>
<th>( \Delta K_{T_{prop}} )</th>
<th>( \Delta K_{T_{duct}} )</th>
<th>( \Delta K_{T_{tot}} )</th>
<th>( \Delta K_Q )</th>
<th>( \Delta \eta_0 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00</td>
<td>0.92%</td>
<td>-1.16%</td>
<td>-0.20%</td>
<td>-5.69%</td>
<td>0.00%</td>
</tr>
<tr>
<td>0.10</td>
<td>1.46%</td>
<td>-2.79%</td>
<td>-1.32%</td>
<td>-5.16%</td>
<td>-0.08%</td>
</tr>
<tr>
<td>0.20</td>
<td>2.97%</td>
<td>-4.89%</td>
<td>-1.92%</td>
<td>-4.37%</td>
<td>1.84%</td>
</tr>
<tr>
<td>0.30</td>
<td>3.73%</td>
<td>-5.81%</td>
<td>-2.09%</td>
<td>-4.96%</td>
<td>0.66%</td>
</tr>
<tr>
<td>0.40</td>
<td>4.94%</td>
<td>-6.19%</td>
<td>-1.75%</td>
<td>-5.39%</td>
<td>1.01%</td>
</tr>
<tr>
<td>0.50</td>
<td>5.26%</td>
<td>-5.98%</td>
<td>-0.82%</td>
<td>-6.12%</td>
<td>3.65%</td>
</tr>
<tr>
<td>0.60</td>
<td>6.10%</td>
<td>-4.62%</td>
<td>1.45%</td>
<td>-7.04%</td>
<td>11.93%</td>
</tr>
</tbody>
</table>

Table 3.6: Relative error between the full RANS calculation of Ka4-70 propeller in 37 duct and the experimental data from the Ka-propeller series. Computed from the dataset of the CFD computations carried out at Wartsila for the CRS work group, [14].
3.5.1. Processing of the chordwise pressure distribution on the duct
The circumferentially averaged pressure distribution on the duct surfaces is useful to compare the full RANS results with the axisymmetric RANS-BEM. In view of the limited data, the mean distribution cannot be computed properly by averaging over all angles, but only few angular steps are considered for two loading conditions (J=0.2 and 0.5). Chordwise distributions are given at 10 angular positions, from 0° to 90° with 10° steps, some of them presented in figure 3.10 (here x/c=0 refers to the nozzle leading edge and x/c=1 to the trailing edge). The wiggles in the pressure are likely related to interpolation issues. In order to compute the mean distribution, a weighted average approach is adopted with the weights obtained by similarity with the case of the 19A duct: the full RANS computations for the duct 19A are again considered and the weights are tuned in such a way that the weighted average using 10 angles is as much similar as possible to the proper average over all angles. The tuned weights are shown in table 3.7 and given in terms of circular sector for which each slice is representative. This means that, for instance, the pressure distribution at 50° is counting for 11° up to the total of 90° (propeller is 4 bladed). Then, it is assumed that the same weights are representative for a good average also in case of the 37 duct and the black dotted distribution shown in figure 3.10 is obtained. This is used in chapter 7 for comparison with potential flow and hybrid calculations.

<table>
<thead>
<tr>
<th>Angle</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>5/90</td>
</tr>
<tr>
<td>10°</td>
<td>10/90</td>
</tr>
<tr>
<td>20°</td>
<td>9/90</td>
</tr>
<tr>
<td>30°</td>
<td>10/90</td>
</tr>
<tr>
<td>40°</td>
<td>11/90</td>
</tr>
<tr>
<td>50°</td>
<td>11/90</td>
</tr>
<tr>
<td>60°</td>
<td>10/90</td>
</tr>
<tr>
<td>70°</td>
<td>9/90</td>
</tr>
<tr>
<td>80°</td>
<td>10/90</td>
</tr>
<tr>
<td>90°</td>
<td>5/90</td>
</tr>
<tr>
<td>Σ weights = 1</td>
<td></td>
</tr>
</tbody>
</table>

Figure 3.10: Chordwise pressure distribution on the duct at several angular location, together with computed mean values. Dataset from [14]. MS: model scale.

Table 3.7: Angular locations of the available slices and weights attributed to compute the mean.

3.6. Summary
The chapter presents the data from full RANS computations and experiments, which are used in this work for validation of the boundary element method and RANS-BEM computations. Experimental open water diagrams are available for both duct geometries, but cavitation observations and pressure measurements are only available for the 19A duct. Furthermore, limited data are available for the full RANS calculations of the Ka4-70 in 37 duct, hence a less complete validation is possible. The expensive 3D calculations provide differences with respect to experimental measurements of 2% at the best, but for certain loading condition and for the more challenging 37-duct geometry, the differences can be up to 7-8%. This is the level of accuracy that can be achieved with a full RANS approach, and therefore these are the values that the potential flow and hybrid calculations are confronted with in the coming chapters.
Procal computations of the Ka4-70 propeller in 19A duct

4.1. Introduction
This chapter presents the results for the Procal computations of the Ka4-70 propeller in the 19A duct. The objective is to determine the optimal settings for Procal and understand the limits of the potential flow package. First, the paneling and the numerical settings are described, followed by the choice for the modified duct geometry. The effect of the hub diameter is then studied and a sensitivity study for the transpiration velocity model implemented in Procal is carried out. Finally, a detailed validation of the pressure distribution on duct and blade surfaces is provided by comparison with the experimental data and the full RANS simulations. The results for advance ratios of \( J = 0.2, 0.5 \) and 0.8 are reported here, while in the Appendix plots for the other loading conditions are provided, with validation data where available.

<table>
<thead>
<tr>
<th>Sections</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.2 Geometry and settings</td>
<td>Paneling of blade-duct-hub-gap surfaces and computational settings</td>
</tr>
<tr>
<td>4.3 Trailing edge geometry</td>
<td>Choice of modified trailing edge for Procal</td>
</tr>
<tr>
<td>4.4 Effect of the hub diameter</td>
<td>Open water diagrams with ( D_{HUB}/D = 0.167 ) and ( D_{HUB}/D = 0.204 )</td>
</tr>
<tr>
<td>4.5 Transpiration velocity model</td>
<td>Sensitivity study for the transpiration model on the duct surface</td>
</tr>
<tr>
<td>4.6 Validation of pressure distributions.</td>
<td>Comparison of Procal results with full RANS</td>
</tr>
</tbody>
</table>

Table 4.1: Outline of the chapter: description of the tasks carried out.

4.2. Geometry and settings
Procal simulations of the Ka4-70 propeller in 19A duct were carried out at MARIN in 2013 by Willemsen and Bosschers\cite{12}\cite{8}. Those studies focused on the grid refinement for the blade and duct surfaces, wake alignment and settings for the boundary layer model and gap flow model. The guidelines for the panel density are followed in this work, resulting in the settings in table 4.3; figure 4.1 shows the panel distribution for the current calculations and table 4.2 lists the details of the geometry and the common settings used for all Procal computations. As concerns the hub diameter, two geometries are used (namely propeller models 7290 and 5979), for which the same panel density on both hub and blade is adopted.
4. Procal computations of the Ka4-70 propeller in 19A duct

<table>
<thead>
<tr>
<th><strong>Propeller</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades $Z$</td>
<td>4</td>
</tr>
<tr>
<td>Propeller models</td>
<td>5979 and 7290</td>
</tr>
<tr>
<td>Diameter $D$</td>
<td>$0.24 m$ (model 5979), 0.21887m (model 7290)</td>
</tr>
<tr>
<td>Hub diameter $D_{HUB}/D$</td>
<td>$0.167$ (model 5979), 0.204 (model 7290)</td>
</tr>
<tr>
<td>Rotation rate $n$</td>
<td>15 rps</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Duct</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Duct model</td>
<td>19A</td>
</tr>
<tr>
<td>Length $L_{duct}$</td>
<td>$0.5 \times D$</td>
</tr>
<tr>
<td>Gap size</td>
<td>1 mm</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Wake model</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Wake model</td>
<td>Iterative wake alignment for the pitch</td>
</tr>
<tr>
<td></td>
<td>Prescribed model for the contraction</td>
</tr>
</tbody>
</table>

Table 4.2: Geometry and wake model settings for the Procal computations of Ka4-70 in 19A duct.

<table>
<thead>
<tr>
<th><strong>Propeller</strong></th>
<th><strong>Spacings</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Blade panels</td>
<td>LE</td>
</tr>
<tr>
<td>1500 (60x25)</td>
<td>0.001</td>
</tr>
<tr>
<td></td>
<td>TE</td>
</tr>
<tr>
<td></td>
<td>0.004</td>
</tr>
<tr>
<td></td>
<td>Root</td>
</tr>
<tr>
<td></td>
<td>0.005</td>
</tr>
<tr>
<td></td>
<td>Tip</td>
</tr>
<tr>
<td></td>
<td>0.006</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Duct</strong></th>
<th><strong>Spacings</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Panels (inner:outer)</td>
<td>LE</td>
</tr>
<tr>
<td>4900 (100x35:40x35)</td>
<td>0.005</td>
</tr>
<tr>
<td></td>
<td>TE</td>
</tr>
<tr>
<td></td>
<td>0.05</td>
</tr>
<tr>
<td></td>
<td>Between blades</td>
</tr>
<tr>
<td></td>
<td>0.04</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Hub</strong></th>
<th><strong>Spacings</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Hub panels</td>
<td>HubU.S.</td>
</tr>
<tr>
<td>800 (80x10)</td>
<td>0.02</td>
</tr>
<tr>
<td></td>
<td>HubT.</td>
</tr>
<tr>
<td></td>
<td>0.02</td>
</tr>
<tr>
<td></td>
<td>HubD.S.</td>
</tr>
<tr>
<td></td>
<td>0.02</td>
</tr>
<tr>
<td></td>
<td>HubN.R.</td>
</tr>
<tr>
<td></td>
<td>0.01</td>
</tr>
</tbody>
</table>

Table 4.3: Blade, duct and hub meshing with values for the spacings.

Figure 4.1: Paneling on the duct, the hub and the blades for Procal simulations of the Ka4-70 propeller in 19A duct.
4.3. Trailing edge geometry

To apply the wake alignment model and the Kutta condition, a sharp trailing edge for the duct has to be defined in Procal. In this study, it is assumed that the effect of different duct geometries does not affect the optimal panel density for the duct surface, nor for the blade. Therefore, the same panel distribution is used for the modified trailing edge geometries presented in this section. With reference to figure 4.2, to define the modified duct geometry, the location of three points is defined: the x and y coordinates of the trailing edge (green point), the location where the modified inner surface fairs into the original geometry (yellow point) and the fairing location for the outer surface (cyan point).

![Figure 4.2: Sketch of the 19A duct geometry and the iterative scheme for the modified trailing edge.](image)

It was shown (Willemsen,[12] 2013) that the axial location of the trailing edge, i.e. the green point in figure 4.2, does not influence the duct loading whereas a different radial position has a large effect on the duct pressure distribution on the duct inner surface, hence on the duct thrust. A modified geometry based on the visualization of the streamlines from a full RANS solution was proposed, which provided satisfactory results[12]. As concerns the fairing location, their effect is secondary compared to the strong variations obtained by changing the radial position of the trailing edge (Mouljin[34], 2014).

In this work, an alternative method to determine an appropriate trailing edge location is investigated, which does not rely on visual observation from RANS calculations. When a symmetric blunt body moves in a viscous fluid, a separation region develops at the back of the body. In a most simplified model, the surface pressure downstream of the detachment point is constant and equal to the pressure within the recirculation region. This consideration is applied to the mean pressure on the duct to determine the location of the trailing edge. A similar method was applied by Kinnas[27], 2013.

The fairing locations are fixed. Also the x coordinate of the trialing edge is fixed at a value of \( x/L_{\text{duct}} = 1.08 \), which means that the duct is extended compared to the original geometry, as in figure 4.2. The idea is to determine the trailing edge radial location such that the pressure difference \( \Delta p = p_1 - p_2 \) in correspondence of the original trailing edge is zero. It is assumed that the pressure difference (in terms of pressure coefficient \( C_{pN} \)) is a function of the trailing edge radial position:

\[
\Delta C_{pN} = C_{pN,\text{outer}} - C_{pN,\text{inner}} = f\left(\frac{y_{te}}{L}\right),
\]

and a secant method is implemented to find the root. The radial position at the i-th iteration is a function of the previous two values of radial position and pressure difference:

\[
\left(\frac{y_{te}}{L}\right)^i = \left(\frac{y_{te}}{L}\right)^{i-1} - \Delta C_{pN}^{i-1} \left(\frac{y_{te}}{L}\right)^{i-2} - \left(\frac{y_{te}}{L}\right)^{i-3}
\]

therefore two initial values need to be prescribed to initialize the iterative scheme. At each step, the duct geometry is different therefore a re-paneling is necessary through Provis in before the next Procal computation takes place. Convergence is reached when the absolute value of the pressure difference falls below a set tolerance.

Several combinations of initial values were tested in a range of \( y_{te}/L \) from 0.015 to 0.055, which correspond to the trailing edge located on the extension of the inner and outer surface respectively. The initial guesses were found not to influence the solution nor the convergence behavior.

Figure 4.3 shows the convergence behavior for \( J=0.5 \) and the initial condition at \( \frac{y_{te}}{L} = 0.020,0.021 \). 4.3a shows also the initial values while in 4.3b the initial values are discarded. The top plots display the radial position as function of iteration number, while the lower ones give the pressure difference. The figures presented
Procal computations of the Ka-70 propeller in 19A duct

4.4. Procal computations of the Ka-70 propeller in 19A duct

here refer to a case with a rather strict tolerance criterion for $\Delta C_{pn}$ of 1E-4 and the maximum number of iteration set to 30. Only five iterations are required to get the pressure difference fall by two order of magnitudes, from roughly 2E-01 to 5E-03, with the radial position increasing to approximately 0.0293. After iteration 5 the behavior becomes oscillatory. This is due to the fact that a detailed information on the flow (i.e. the pressure at two points) is used for this scheme, hence small changes in geometry can result in relatively different pressure values, affecting the accuracy of the secant scheme. Nevertheless, the radial position does not change a lot after iteration 15, despite the oscillation in $\Delta C_{PN}$. Then, it can be concluded that, for practical purpose, a tolerance of 1E-3 is recommended with a safe maximum number of iteration set to 15.

Table 4.4 lists the trailing edge position obtained for the different loading conditions. Different initial conditions or a stricter tolerance criterion were found to influence only the fourth decimal, which is truncated in the table. For the lower J, the scheme diverges. The divergence is related to the poor behavior of the wake alignment in Procal (or, in certain cases, even in the IPKC). Indeed, when the Procal solution diverges at the i-th iteration, the interpolated pressure difference becomes much larger, leading to failure in computing the slope at the i-th+1 iteration hence the divergence of the scheme.

<table>
<thead>
<tr>
<th>Advance ratio</th>
<th>Radial position T.E.</th>
</tr>
</thead>
<tbody>
<tr>
<td>J</td>
<td>$y/L$</td>
</tr>
<tr>
<td>0.2</td>
<td>0.029*</td>
</tr>
<tr>
<td>0.3</td>
<td>0.029*</td>
</tr>
<tr>
<td>0.4</td>
<td>0.028</td>
</tr>
<tr>
<td>0.5</td>
<td>0.029</td>
</tr>
<tr>
<td>0.6</td>
<td>0.031</td>
</tr>
<tr>
<td>0.7</td>
<td>0.032</td>
</tr>
<tr>
<td>0.8</td>
<td>0.034</td>
</tr>
<tr>
<td>Formerly[12] (all J)</td>
<td>0.031</td>
</tr>
</tbody>
</table>

Table 4.4: Radial position of the trailing edge obtained with the iterative scheme for the pressure difference. * not converged.

In addition, figure 4.5 shows for J=0.5 the mean duct pressure distribution and the blade axial force. The distributions for the initial guesses (here at $y/L = 0.020, 0.021$, both on the visual extension of the inner surface) are shown in blue, whereas the converged solution in green. The duct pressure distribution differ mostly at the duct inner surface. At the location of the original trailing edge (the red vertical line) the pressure at the inner and outer side differs by approximately 2E-01 for the initial guesses, whereas it drops to zero for the converged solution as expected. The wiggles for the inner surface are due to a poor mesh distribution on the duct close to the blade tip trailing edge. Even though the pressure at the expected separation point is forced to be equal at the two sides of the duct, there is a non-physical pressure recovery towards the modified trail-
4.3. Trailing edge geometry

Figure 4.5: Mean duct pressure distribution and axial force for the initial guesses and the converged solution, J=0.5.

The trailing edge, at coordinate x/c larger than one. The loading on the duct is smaller with the trailing edge on the extension of the inner surface, hence the acceleration of the flow is less pronounced. Coherently, the blade axial force is large closer to the tip of the blade, because the less induced velocity result in a larger blade angle of attack, so a higher loading.

Figure 4.6: Open water diagram and radial force for the initial guess at $\frac{y_{TE}}{L_{duct}} = 0.020$ and the converged solution.

Figure 4.6 displays the open water diagram and the radial force for the initial condition $\frac{y_{TE}}{L_{duct}} = 0.020$ and the converged solution (except at J=0.2 and 0.3 when the scheme did not converge, even though the solution at the last iteration is still reasonable, hence it is here reported). As noted by Bosschers et al.[8],2014 the trailing edge position hardly affect the duct thrust, but largely influences the radial force acting on the duct, the propeller thrust and torque. For the converged solution, the propeller thrust and torque well match with the experimental data.

Even though, as in table 4.4, the scheme produces different trailing edge position for the advance ratios, it is not convenient to use different geometries for each J. Therefore, a choice is made for a trailing edge position which provides the best compromise. In view of the design advance ratio of J=0.55 and the results for the other loading condition, a radial position of $\frac{y_{TE}}{L} = 0.031$ is chosen and used for the reference computations in the following of this work.

Figure 4.7 shows the open water diagram for the reference trailing edge geometry and for the computed geometry for each J in table 4.4. There is a maximum of 8% difference, occurring for the propeller thrust and torque at J=0.8. This discrepancy is accepted and a single trailing edge position, as in figure 4.8 will be used for all J.

In conclusion, the method based on imposing an equal pressure at the duct surface is a good alternative for choosing the radial position of the trailing edge to the streamline observation from RANS or visual guess.
However, the scheme is as stable as the Procal iterative wake alignment, so issues arise at the lowest advance ratios. Furthermore, it was shown that, although the computed position changes with $J$, choosing one geometry gives satisfactory results throughout the open water range. Further development of the current scheme is possible. A possibility would be to retain the original length of the duct and prescribe the pressure-equality condition at a more upstream station, whose location is to be studied. This would be a straightforward investigation using the current implementation. More sophisticated methods are also available in literature. Examples are the 2 degrees of freedom model ($x$ and $y$ coordinate of trailing edge) and the cavity-like model implemented by Pan, [35], 2009. Finally, the interpolation of duct loading required in this scheme is predictably grid-dependent, therefore a grid sensitivity study is also recommended for future work to assess whether the resultant trailing edge radial position would be grid dependent.

### 4.4. Effect of hub diameter

For the experimental tests on the Ka4-70 propeller in a 19A nozzle (Foeth[10], 2012), a hub diameter of $D_{hub}/D = 0.204$ was used, while for the RANS computations the hub diameter is $D_{hub}/D = 0.167$ (Rijpkema[11], 2012) in accordance with the original geometry. The two propeller geometries are reported in the appendix: model 5979 has $D_{hub}/D = 0.167$ while model 7290 has $D_{hub}/D = 0.204$.

The effect of a different hub diameter is investigated in Procal using the two hubs. Figure 4.9 reports the open water diagram and the radial force. As concerns the radial force, it was not measured during the experimental tests, therefore the full RANS solution is here reported. Table 4.5 also lists the loadings for two operating conditions, high loading and close to the design condition ($J=0.55$). The relative difference for the quantity $\phi$ with respect to the two hub diameters is defined as:

$$\Delta \phi = \left| \frac{\phi_1 - \phi_2}{\max(\phi_1, \phi_2)} \right| \times 100 \ [%] \ (4.3)$$

The propeller thrust coefficient is the most affected quantity, with a maximum relative change of 3% at $J=0.5$. This was expected, since the blade is bigger for the case with small hub. The duct thrust is hardly affected, so the total thrust coefficient $K_T$ differs of roughly 2%. The radial force acting on the duct is approximately 2% larger (in absolute value) for the large hub. This is explained by considering the blockage effect induced by the larger hub. The smaller flow section induces larger acceleration and so, in a potential analysis, a smaller pressure at the inner side of the duct, which results in a larger radial force. The larger flow acceleration also leads to a decrease in the blade angle of attack, which further contributes to the reduction in $K_{TP}$. When comparing the Procal solution with the experimental results, for both geometries there is an overprediction of the duct thrust at large advance ratios, when the viscous effects lead to flow separation at the outer side of the duct (Bosschers et al. [8], 2014), which is not modeled in the potential flow package. This results in the over-prediction of the open water efficiency. This will be significantly improved by matching the duct loading in the RANS-BEM coupling through the transpiration velocity model (section 2.4.2).
4.5. Transpiration velocity model

As explained in chapter 2, a transpiration velocity model is implemented in Procal for the duct panels. The effect of the boundary layer on the duct is modeled by blowing sources whose strength is given by the impermeability condition with the addition of a transpiration velocity normal to the duct surface. This results in a change of the duct camber line, hence the duct loading. The model assumes that the boundary layer grows as a cubic of the non-dimensional arc length, up to a maximum of \( \delta^*_{TE} \), the displacement thickness at the trailing edge [36]. The transpiration velocity model is used to match the duct force in the RANS-BEM coupling, but it can also be used for the pure BEM simulations. In this case, the user has to prescribe a fixed value of \( \delta^*_{TE} \) given as a fraction of the duct length \( L_{duct} \), together with the location where the transpiration velocity model is applied. In this study, the effect of an increasing value of the boundary layer displacement thickness is investigated, as well as two options for the location of the blowing sources: on the whole duct inner surface or on the inner surface downstream of the blades only. Table 4.6 gives an overview of the settings used for the computations.

As a conclusion, a difference of maximum 3% in the prediction of propeller loading is accepted for comparing the potential flow solution with the validation material (full RANS solution and experimental data). Furthermore, the effect of a different hub geometry is negligible compared to the effect of the Procal settings for wake alignment, boundary layer correction or transpiration velocity model (see Willemson [12], 2013 and the following sections.) Therefore, the case with small hub (in accordance with the full RANS computations) is chosen as the reference Procal computation for this test case, unless differently specified.

4.5. Transpiration velocity model

As explained in chapter 2, a transpiration velocity model is implemented in Procal for the duct panels. The effect of the boundary layer on the duct is modeled by blowing sources whose strength is given by the impermeability condition with the addition of a transpiration velocity normal to the duct surface. This results in a change of the duct camber line, hence the duct loading. The model assumes that the boundary layer grows as a cubic of the non-dimensional arc length, up to a maximum of \( \delta^*_{TE} \), the displacement thickness at the trailing edge [36]. The transpiration velocity model is used to match the duct force in the RANS-BEM coupling, but it can also be used for the pure BEM simulations. In this case, the user has to prescribe a fixed value of \( \delta^*_{TE} \) given as a fraction of the duct length \( L_{duct} \), together with the location where the transpiration velocity model is applied. In this study, the effect of an increasing value of the boundary layer displacement thickness is investigated, as well as two options for the location of the blowing sources: on the whole duct inner surface or on the inner surface downstream of the blades only. Table 4.6 gives an overview of the settings used for the computations.

As a conclusion, a difference of maximum 3% in the prediction of propeller loading is accepted for comparing the potential flow solution with the validation material (full RANS solution and experimental data). Furthermore, the effect of a different hub geometry is negligible compared to the effect of the Procal settings for wake alignment, boundary layer correction or transpiration velocity model (see Willemson [12], 2013 and the following sections.) Therefore, the case with small hub (in accordance with the full RANS computations) is chosen as the reference Procal computation for this test case, unless differently specified.

### Table 4.5: Open water data for the geometries with small and large hub diameter.

<table>
<thead>
<tr>
<th></th>
<th>Small hub</th>
<th>Large hub</th>
<th>Relative difference between hubs(%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( K_{TP} )</td>
<td>2.60E-01</td>
<td>2.54E-01</td>
<td>2.08</td>
</tr>
<tr>
<td>( K_{TD} )</td>
<td>1.90E-01</td>
<td>1.91E-01</td>
<td>0.47</td>
</tr>
<tr>
<td>( K_T )</td>
<td>4.50E-01</td>
<td>4.45E-01</td>
<td>1.00</td>
</tr>
<tr>
<td>( 10K_Q )</td>
<td>4.42E-01</td>
<td>4.33E-01</td>
<td>1.83</td>
</tr>
<tr>
<td>( F_r )</td>
<td>-5.02E-01</td>
<td>-5.15E-01</td>
<td>2.45</td>
</tr>
<tr>
<td>( J=0.2 )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( K_{TP} )</td>
<td>1.92E-01</td>
<td>1.87E-01</td>
<td>3.07</td>
</tr>
<tr>
<td>( K_{TD} )</td>
<td>7.03E-02</td>
<td>7.03E-02</td>
<td>0.11</td>
</tr>
<tr>
<td>( K_T )</td>
<td>2.63E-01</td>
<td>2.57E-01</td>
<td>2.22</td>
</tr>
<tr>
<td>( 10K_Q )</td>
<td>3.47E-01</td>
<td>3.39E-01</td>
<td>2.33</td>
</tr>
<tr>
<td>( F_r )</td>
<td>-3.99E-01</td>
<td>-4.09E-01</td>
<td>2.59</td>
</tr>
<tr>
<td>( J=0.5 )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( K_{TP} )</td>
<td>1.92E-01</td>
<td>1.87E-01</td>
<td>3.07</td>
</tr>
<tr>
<td>( K_{TD} )</td>
<td>7.03E-02</td>
<td>7.03E-02</td>
<td>0.11</td>
</tr>
<tr>
<td>( K_T )</td>
<td>2.63E-01</td>
<td>2.57E-01</td>
<td>2.22</td>
</tr>
<tr>
<td>( 10K_Q )</td>
<td>3.47E-01</td>
<td>3.39E-01</td>
<td>2.33</td>
</tr>
<tr>
<td>( F_r )</td>
<td>-3.99E-01</td>
<td>-4.09E-01</td>
<td>2.59</td>
</tr>
</tbody>
</table>

Figure 4.9: Comparison of the open water loading for the geometries with small and large hub.

As a conclusion, a difference of maximum 3% in the prediction of propeller loading is accepted for comparing the potential flow solution with the validation material (full RANS solution and experimental data). Furthermore, the effect of a different hub geometry is negligible compared to the effect of the Procal settings for wake alignment, boundary layer correction or transpiration velocity model (see Willemson [12], 2013 and the following sections.) Therefore, the case with small hub (in accordance with the full RANS computations) is chosen as the reference Procal computation for this test case, unless differently specified.
4.5.1. Location of transpiration velocity

Figures 4.10 show the effect of the two locations for the transpiration velocity model. They refer to a fixed value of the boundary layer displacement thickness at the trailing edge of 2% (left plot) and 4% (right plot) of the duct length. The duct thrust is barely affected. As expected, with a thicker boundary layer, there is a larger discrepancy in propeller thrust and torque between the two models because the transpiration velocity is larger. Furthermore, for both values of $\delta_{TE}/L_{duct}$, the solutions differ the most for the large advance ratios. This is because the transpiration velocity gets larger with larger ship velocity (as in equation (2.83)), hence the effect of a different location for transpiration is also more evident. Moreover, when the boundary layer displacement thickness at the trailing edge gets larger, the solution with transpiration on the whole inner side (the source model 3) provides a significant overestimation of propeller thrust and torque, which is also observed for the source model 2 but to a less extent. This is the reason why the model with transpiration on the duct inner surface downstream of the blade is chosen for the following of this study.

![Figure 4.10: Open water diagrams comparing transpiration on the whole inner side (source model 3) and transpiration from blade TE to duct TE (source model 2).](image)

4.5.2. Boundary layer displacement thickness at the trailing edge

For the selected transpiration velocity location on the aft duct inner surface, a sensitivity study is carried out for the user-prescribed value of the displacement thickness. As in table 4.6, the value for $\delta_{TE}$ ranges from 0.5% to 8% of the duct length. Figure 4.11 shows how the propeller thrust and the duct radial force changes with the boundary later thickness for advance ratios of 0.3,0.5 and 0.8. The behavior is linear; $K_{TP}$ increases, whereas $F_{rd}$ decreases in absolute value, with also positive values for $J=0.8$ and $\delta_{TE}/L_{duct} > 0.02$. The larger boundary layer thickness results in a smaller duct loading which relates to a smaller induced velocity, such that the blade angle of attack increases and so the blade loading gets larger. This effect is the same as observed in section 4.3 for the Procal trailing edge geometry.

To quantify the sensitivity of duct and propeller loading to the settings in transpiration model, the relative difference in propeller thrust, duct thrust and duct radial force with reference to the case without transpiration is computed and plotted in figure 4.13 for $J=0.3,0.5$. At the x axis, the relative difference in radial force is presented percentage-wise, while at the y-axis both the propeller thrust and the duct thrust are given. The relative differences are larger for $J=0.5$ than $J=0.3$, but in both cases the radial force is the most affected quantity (the lines all lies below the bisector). $F_{rd}$ varies roughly 1.5 times the propeller thrust coefficient, and 4 to 5 times the duct thrust coefficient.

The radial force largely changes with the boundary layer thickness. Figure 4.12 shows that a good match with

<table>
<thead>
<tr>
<th>Key Name</th>
<th>Description</th>
<th>Options used</th>
</tr>
</thead>
<tbody>
<tr>
<td>source_correction_factor</td>
<td>Boundary layer thickness at the trailing edge as fraction of the duct length</td>
<td>$\delta_{TE}/L_{duct} = 0, 0.005,0.01,0.02,0.04,0.06,0.08.$</td>
</tr>
<tr>
<td>source_correction_model</td>
<td>Location where the transpiration velocity is applied</td>
<td>2: from blade trailing edge to duct trailing edge,</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3: the whole duct inner surface.</td>
</tr>
</tbody>
</table>

Table 4.6: Settings for the transpiration velocity model. Key name for Procal control file, description and options used for computations.
the RANS solution is obtained when $\delta_{TE}^*/L_{duct} = 0.04$. It is remarkable that the radial force is well matched throughout the open water diagram with the same value of $\delta_{TE}^*$ whereas the open water diagram on the right of figure 4.10 shows that the duct thrust is over-predicted for $J>0.5$. This difference in the two components of the duct loading is related to the pressure distribution on the duct surface as explained in the following section. Nevertheless, it is useful to bear in mind that the Procal solution which best matches the duct loading, given the limits of the potential flow code, is the one for a trailing edge displacement thickness of 4% the duct length. This information is used to choose a suitable Procal computation for validation of the pressure distribution on the duct in section 4.6.

Figure 4.11: Effect on propeller thrust (left axis) and duct radial force (right axis) of variation of the displacement thickness $\delta_{TE}^*$.

Figure 4.12: Radial force, Procal solution with $\delta_{TE}^*/L_{duct} = 0.005, 0.04, 0.08$ and ReFresco solution. For $\delta_{TE}^*/L_{duct} = 0.005$ and $J=0.2$ convergence problems were observed.

Figure 4.13: Relative difference in the loading with an increasing $\delta_{TE}^*$ taking the solution without transpiration as reference. Percentages give the relative difference in the radial force at the x-axis and the relative difference in thrust (both propeller and duct thrust) at y axis. The duct radial force varies roughly 1.5 times the propeller thrust and 4.5 times the duct thrust.
4.5.3. Influence on the pressure distribution at the duct surface
The effect of the transpiration model on the pressure distribution on the duct is investigated. Figures 4.14 and 4.15 show the chordwise pressure distribution (c is the chordlength) for three Procal solutions with increasing δ∗ and the full RANS solution for an advance ratio of 0.5 and 0.8 respectively. Two plots are given per each condition, at blade angles of 0° (12 o’clock position on top of the blade) and 45° (in between the blades for the 4 bladed propeller). The y axis gives the pressure coefficient, and the x axis the dimensionless chordwise position. The latter is larger than 1 because for Procal the trailing edge geometry is modified with the trailing edge point located at a more downstream position (see section 4.3).

When J=0.5 in figures 4.14 the pressure distribution on the outer duct surface is well matched with the RANS solution and is not affected by a different boundary layer thickness. This was predictable because the chosen location for the transpiration model is on the duct inner side from the blade trailing edge to the duct trailing edge. As concerns the pressure on the inner surface, there is a jump at x/c ≈ 0.5 at 0° blade angle where there is an abrupt change in pressure related to the suction and pressure sides on the blade. There is then an increase in pressure with larger value of δ∗ which is coherent with the fact that the negative radial force diminishes with larger boundary layer thickness. Furthermore, the Procal solutions under-predict the pressure, especially upstream of the blade (at x/c < 0.5) for 0° blade angle. Downstream of the blade, the RANS solution shows pressure peaks due to the tip vortices convected in the wake, which are not present in potential flow. These two effects of under-prediction upstream of the blade and lack of tip vortex downstream of the blade balance each other, such that the duct radial force predicted by Procal with δ∗/L duct = 0.04 matches with the full RANS solution at J=0.5 (figure 4.12) and the duct thrust is reasonably close to the experimental value for J=0.5, as in the right diagram of figure 4.10.

Moreover, the underestimation of pressure at 0° blade angle and x/c < 0.5 suggests that a better pressure distribution could be obtained with a transpiration velocity throughout the whole duct inner surface (i.e. the source_correction_model=3), but this was observed to lead to an excessive pressure downstream of the blade. Since the current method is based on a third power boundary layer growth

δ∗ = δ∗TE 3,

a better match of the duct pressures in design condition could be obtained by applying the transpiration velocity at the whole inner surface and lowering the exponent in the power law for the boundary layer growth. As concerns the higher advance ratio J=0.8, the pressure distribution from Procal and ReFRESCO differs both at the duct inner and outer surface. In figure 4.15 at a blade angle of 0° the distribution at the inner side is well matched with the RANS solution up to x/c = 0.2, whereas for a blade angle of 45° the distribution for the Procal solution with δ∗/L duct = 0.04 shows small differences with the RANS, with exception of the pressure peaks due to the tip vortex. Differently, the pressure distribution at the outer surface significantly differ between the potential flow solutions and the full RANS. At both blade angle, there is an under-prediction of the pressure in Procal at small x/c, followed by an non-physical pressure recovery downstream. This is because at J=0.8 the flow detaches close to the duct leading edge and a large recirculation region is observed on the outer side, Rijpkema[11], 2012. These phenomena are not modeled in Procal, where the flow remains attached.

By looking at the duct thrust in the right diagram of figure 4.10 and the plot of the radial force 4.12, it appears that for J ≤ 0.5 both the duct thrust and the radial force are well matched with the experimental data and the RANS solution respectively. This is not the case for large advance ratios, when the radial force is still well predicted but the duct thrust shows an overestimation. For the RANS solution, the contribution of the shear stress to the duct thrust is around 4% of the total X-force, while for the radial force the contribution is less than 2%. Therefore, the differences in duct loading between Procal and ReFRESCO can be explained by looking at the pressures only. Sketch 4.16 gives explanation of why the two components of loading behave differently at different J. When J=0.5, the mispredictions in pressures at the inner surface from Procal balance out while for J=0.8 the x components of the force is overestimated because the normal vector in the region where the pressure does not match has always a positive X component.
4.5. Transpiration velocity model

Figure 4.14: Chordwise pressure distribution on the duct at blade angle of 0° and 45° with increasing value of displacement thickness at the trailing edge. Procal computations and full RANS results for J=0.5.

Figure 4.15: Chordwise pressure distribution on the duct at blade angle of 0° and 45° with increasing value of displacement thickness at the trailing edge. Procal computations and full RANS results for J=0.8.

Figure 4.16: Sketch of the regions where the pressure is mispredicted in the Procal solution. The red arrows give the vectors \((p_{\text{RANS}} - p_{\text{BEM}})\hat{r}\). For J=0.5 the misprediction in pressures balance each other out. For J=0.8 the duct thrust is overestimated.
4.6. Validation: pressure on the duct and the blade surfaces

In this section the solution from Procal is validated against the full RANS and the experimental data for what concerns the pressure distributions both on the blade surface and on the duct.

Figure 6.8 shows the $C_{PN}$ distribution on the blade for the Procal reference solution and the full RANS solution for $J=0.5$. The contours for the other advance ratios are given in the Appendix. At the pressure side, the distribution differs between the BEM and RANS towards the trailing edge and close to the tip. At the trailing edge and large radii there is a region (in light blue) of low pressure in the RANS solution and also at smaller radii the Procal solution shows a larger region (light green area) close to the trailing edge where the pressure is overestimated. This difference is expected to be related to the blade trailing edge geometry adopted in the RANS computations ([11]). At the leading edge, the pressure contours resemble, except close to the tip where the RANS solution exhibits a region of lower pressure.

Regarding the suction side, the distribution is similar up to a radius of $r/R \approx 0.5$. At large radii Procal underpredicts the pressure close to the leading edge and over-predicts it at the trailing edge. Both mis-predictions are more evident at high loading condition (contour plots for $J=0.2$ to 0.8 are reported in the Appendix). Furthermore, in the RANS solution around the mid-chord position at the tip, there is a region of low pressure related to the tip leakage vortex which is formed in the gap between blade and nozzle. The tip leakage vortex is stronger for higher loading conditions, hence the low pressure area is also larger for small $J$. X-Y plots of the pressure at the blade will be presented together with the RANS-BEM result in chapter 5.

![Blade pressure contours for the Procal reference solution and the full RANS solution, $J=0.5$.](image)

The pressure distribution on the nozzle is now validated against the full RANS data and the experimental measurement. As observed in the previous section, the best prediction of the duct loadings (thrust and radial force) is obtained by modeling the boundary layer with a displacement thickness at the trailing edge.
of 4% the duct length, therefore the Procal solution for such setting is here considered. The angular pressure distribution on the duct is obtained by interpolating the duct pressure at four stations at constant X, as shown in figure 4.18 where also the blade tip section is plotted in black. The duct is sliced at \( x/R = 0.25, 0.075, -0.075, -0.3 \) which correspond to the locations where the pressure gauges in the experimental data were fitted, Foeth[10]2013. The nozzle leading edge is located at \( x/R = 0.5 \) whereas the trailing edge is at \( x/R = -0.5 \), while for Procal computations the trailing edge was extended as previously explained. Advance ratios of \( J=0.5 \) and \( J=0.2 \) were tested during the experiments. Figure 6.8 shows the plots for \( J=0.5 \) while the \( J=0.2 \) case is included in the Appendix. For the experimental data, the blade rate ensemble average of the pressure measurements is used, with a cavitation number of \( \sigma = 1.7 \) corresponding to the atmospheric pressure in the towing tank. A phase correction of the pressure signal is applied [10], which results in a 12° correction based on the angular pressure distribution at \( x/R = 0.25 \) and set for all \( x/R \) slices. The distribution has a period of 90° because the propeller has 4 blades. At the duct outer surface, the pressure oscillates with the same period, but such pattern is not visible because of the axis scale used in the plots.

Upstream of the blade (the top left plot) the angular distribution confirms that the potential flow solution under-estimates the pressure. There is a good match between the RANS solution and the experimental data, given the aforementioned phase correction. At \( x/R = 0.075 \) there is a jump related to the pressure and suction sides of the blade. The Procal solution overestimates the pressure peaks, even though the phase is correct. The full RANS solution predicts a larger pressure than the experimental data with a constant shift throughout the circumferential position. Furthermore, in the experimental data there are secondary pressure peaks at a blade angle around 45° which are not present in the ReFRESCO solution (they are zoomed in the blue circle). Further downstream, at \( x/R = -0.075 \), the jump in pressure is larger in the experimental data than in the full RANS solution, which is itself larger than in the Procal solution. The full RANS solution shows a second, wider peak at blade angle of 50°. This is caused by the tip leakage vortex that detaches towards the blade leading edge and is convected downstream. Noticeably, the experimental data does not exhibit this pattern, therefore a further investigation on the detachment location and pitch of the tip leakage vortex is done in the following of this work. The secondary peak noted at \( x/R = 0.075 \) in the experimental data is also observed at \( x/R = -0.075 \) at a smaller angular position (blue circle in the figure). Regarding the most downstream location at \( x/R = -0.3 \), the pressure from Procal is almost constant, while the experimental and full RANS solution exhibit peaks related to the tip/tip leakage vortices in the propeller wake. The peaks show a phase shift, hence the pitch of the vortex is different.

Overall, Procal is not able to predict correctly the pressure on the duct inner side where the viscous effects are dominant in the flow. In the gap between blade and nozzle the flow is driven by the finite pressure difference between the blade sides, and the shear layer results in vortex roll up and detachment of the tip leakage vortex. These phenomena are not modeled in the potential flow code\(^1\) and largely influence the pressure distribution at \( x/R = -0.075, x/R = 0.075 \) as well as the pressure peaks at \( x/R = -0.3 \).

---

\(^{1}\)In this work the gap model based on discharge coefficient \( C_Q \) for the gap flow has not been enforced. In [34] it was proved that the effect of such model is small, and only affects the pressure distribution on the blade surface at large radii. It is then concluded that its effect is negligible compared to the settings for the trailing edge geometry and the boundary layer displacement thickness, hence a closed-gap model was chosen.
4.7. Summary

In conclusion from the Procal computations, the iterative scheme for determining the trailing edge radial position based on a pressure-equality criterion is shown to be suitable without the need of prior knowledge on the flow behavior (streamlines). Furthermore, the change in the hub diameter hardly affects the duct and propeller forces. Differently, the boundary layer displacement thickness for the transpiration velocity has a large effect on the radial force, less on the total thrust. The sensitivity study to trailing edge displacement thickness shows that the pressure distribution at the duct is sensitive to this parameter and there is a balancing effect for the duct loading such that the integral quantities match with the validation data at design condition, but not at large $J$. Such consideration will turn useful when analyzing the RANS-BEM method (chapter 5) which relies on the transpiration model to match the duct force iteratively. Finally, from the validation task it appears that not only the potential flow package mispredicts the pressures wherever vortices are present, but also there are differences between the full RANS solution and the experimental results, hence a closer look is taken at the gap flow and tip leakage vortex behavior in chapter 8.
ReFRESCO-Procal computations of the Ka4-70 propeller in 19A duct

5.1. Introduction
In this chapter the results for the RANS-BEM calculations of the Ka4-70 propeller in 19A duct are presented. The coupling procedure is based on matching iteratively the duct radial force between Procal and ReFRESCO. A total of 20 coupling steps are necessary to match the radial force; in this case the solution is also referred to as coupled solution. However, for the grid refinement study or sensitivity analysis to the interpolation settings, it is convenient to provide calculations with 1 coupling step only; if that is the case, only one Procal calculation takes place and the propeller force is interpolated once to the RANS grid. At the current stage of development, the RANS-BEM coupling of the propeller and the duct is not available for in-behind condition. Therefore, the effective wake for Procal is the uniform inflow and the extrapolation of the effective wake (as presented for instance in [9]) is not an issue.

As in the previous chapter, two propeller geometries (model 5979 and 7290) are tackled, which differ in the hub diameter. Table 5.1 lists the sections in this chapter together with the results presented and the geometry used for each task. For the $D_{hub}/D = 0.167$ propeller only one grid is used, with grid density as suggested by the work of Willemsen[12] and Bosschers[8]. For the $D_{hub}/D = 0.204$ the grid refinement study is carried out and a more detailed analysis of the flow is performed. Focus in this chapter is given to the loading conditions $J = 0.2,0.5$ and $0.8$, while the results for the other loading conditions are included in the Appendix.

<table>
<thead>
<tr>
<th>Sections</th>
<th>$D_{hub}/D = 0.167$ Prop model 5979</th>
<th>$D_{hub}/D = 0.204$ Prop model 7290</th>
<th>Analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.2 Interpolation scheme</td>
<td>•</td>
<td></td>
<td>Recursive step in Volume Intersection(VI).</td>
</tr>
<tr>
<td>5.3 Grid selection</td>
<td></td>
<td>•</td>
<td>Boundary layer modeling and grid refinement</td>
</tr>
<tr>
<td>5.4 Effect of hub geometry</td>
<td>•</td>
<td>•</td>
<td>Open water diagrams</td>
</tr>
<tr>
<td>5.5 Comparison with BEM</td>
<td>•</td>
<td>•</td>
<td>Loadings and radial force</td>
</tr>
<tr>
<td>5.6 Pressure distributions</td>
<td></td>
<td>•</td>
<td>Validation against full RANS</td>
</tr>
<tr>
<td>5.7 Contour plots</td>
<td></td>
<td>•</td>
<td>Velocity, pressure, vorticity and body forces</td>
</tr>
</tbody>
</table>

Table 5.1: Outline of the chapter with a list of the tasks carried out and the propeller geometry used.

5.2. Interpolation method
As presented in section 2.4, the RANS-BEM procedure requires the interpolation of the propeller force to the RANS mesh. Two methods are used at MARIN: the (Swept Volume Interpolation (SVI)) and the (Volume Intersection (VI)) methods. In [9] the methods are compared for the RANS-BEM coupling of two ship hull-propeller systems, whereas in [12] the effect of the interpolation method is studied for one case of duct-propeller system. The swept volume method is experienced to result in overestimation of the pressure peaks at the blade leading edge, and a poor grid convergence behavior. Therefore, it is decided to use the VI method in this study. It makes use of a recursive algorithm for intersection of the volume of the RANS cells with the...
volume swept by the BEM panels in one time step (so-called panel volumes). The larger the number of recursive steps set by the user, the more precise is the interpolation of propeller force. A preliminary sensitivity study is carried out to determine the proper number of recursive steps, for an advance ratio of \( J = 0.5 \). Table 5.2 provides the computational time and the percentages of panel volume that is interpolated for the range of recursive steps investigated. The interpolated volume is also presented in figure 5.1a. For these computations, 1 coupling step was set; the main settings and characteristics are provided in table 5.3. The volume interpolated increases significantly when comparing the cases with 4 steps (47% of the propeller swept volume) to the case with 10 time steps (99% of the swept volume). However, the CPU time also increases considerably, up to unfeasible duration of over 100 hours. A good combination of accuracy in the interpolation and feasible computation time is obtained with \( \text{maxRefLevels} = 6 \) to 7. Figure 5.1b shows the relative differences in \( K_{TD} \) and \( F_{RD} \) when the case with the largest number of recursive step is taken as the reference value. With 6 and 7 refinement steps there is a relative difference of roughly 2% and 1% respectively in both the duct thrust and radial force. Therefore, in view of the CPU time and the accuracy of the interpolation, a value of 7 refinement steps is used in the following computations.

<table>
<thead>
<tr>
<th>Recursive steps ((\text{maxRefLevels}))</th>
<th>CPU time ([\text{h:mm:ss}])</th>
<th>Interpolated Volume (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>2:43:03</td>
<td>47</td>
</tr>
<tr>
<td>5</td>
<td>4:01:21</td>
<td>71</td>
</tr>
<tr>
<td>6</td>
<td>5:15:13</td>
<td>85</td>
</tr>
<tr>
<td>7</td>
<td>5:46:54</td>
<td>92</td>
</tr>
<tr>
<td>8</td>
<td>12:52:45</td>
<td>95</td>
</tr>
<tr>
<td>9</td>
<td>42:04:01</td>
<td>97</td>
</tr>
<tr>
<td>10</td>
<td>157:21:23</td>
<td>99</td>
</tr>
</tbody>
</table>

Table 5.2: CPU time on 32 cores 2.4GHz and interpolated volume with increasing number of recursive steps.

<table>
<thead>
<tr>
<th>Grid characteristics</th>
</tr>
</thead>
<tbody>
<tr>
<td>Advance ratio ( J )</td>
</tr>
<tr>
<td>Coupling steps</td>
</tr>
<tr>
<td>Hub geometry</td>
</tr>
<tr>
<td>Hub boundary</td>
</tr>
<tr>
<td>( D_{HUB}/D )</td>
</tr>
<tr>
<td>Number cells</td>
</tr>
<tr>
<td>Convective scheme</td>
</tr>
</tbody>
</table>

Table 5.3: Detail of the RANS grid and the geometry used in the sensitivity to the \( \text{maxRefLevels} \) parameter.

Furthermore, a study of the order of convergence of the duct loading with an increasing number of recursive steps is carried out. The convergence index is determined in a similar fashion as in the theory for the grid refinement study. Figures 5.2 show the fit for \( K_{TD} \) and \( F_{RD} \), together with error bars for \( \epsilon \). The observed order is around 5 and 4.5 for the thrust and radial force respectively. This large order is related to the increase of

\[ 1 \] A new and faster user-code routine is under development at MARIN for the enforcement of the Volume Intersection method. For the new routine, the user-set parameter \( \text{maxRefLevels} \) is decreased of one unity. This means that a value of \( \text{maxRefLevels} = 4 \) in the current study will correspond to \( \text{maxRefLevels} = 3 \) in the new routine.
the volume interpolated from the BEM panels to the RANS grid. With the chosen setting of $\text{maxRefLevels}=7$ there is an error of 1.44\% for $K_{TD}$ and 1.54\% for $F_{RD}$, both considered acceptable. This investigation was carried out for one grid only, with the geometry and main settings as in table 5.3. The order of convergence is expected to be grid-dependent; therefore, it is recommended for future work to evaluate also the effect of a grid refinement/coarsening both in Procal paneling and ReFRESCO 3D mesh.

5.3. Grid selection

For the propeller geometry with $D_{Hub}/D = 0.204$ 18 grids were generated, with a combination of three boundary layer resolution for the hub wall and 6 grid densities. In this section, the choice of the grid resolution and refinement for the RANS part of the hybrid method is presented. First of all, the boundary layer resolution for the hub wall is selected, followed by the grid refinement study. Figure 5.3 shows the geometry and the gridding on the duct and hub surfaces, as well as the body force distribution at a plane located at $x/R=0$.

![Figure 5.2: Convergence of the duct thrust and duct radial force with an increasing number of refinement steps in the VI scheme for interpolation of propeller loading.](image)

![Figure 5.3: Grid for the RANS part of the hybrid approach showing the surface mesh density and the body force distribution (the grid has 3.5M cells).](image)
5.3.1. Grid kind
Three boundary layer resolutions for the hub wall are tested: fully-resolved boundary layer, wall-function approximation for the viscous sublayer and free-slip condition at wall. The comparison of the three grid kinds is made for an advance ratio \( J=0.5 \) and for grids of approximately 3 million cells; simulations with 20 RANS-BEM coupling steps are performed because it is relevant to check the difference in CPU time for the fully-coupled solution. Table 5.4 reports the grid settings and resulting duct loading. Both \( K_{TD} \) and \( F_{RD} \) show a maximum difference of less than 1\% between the three grid kinds, with the larger difference occurring for the grid with wall function. The CPU time increases of 10\% from the fastest simulation (with wall function) to the slowest (resolved boundary layer), but it remains within a reasonable duration. In view of the feasible CPU time and the better flow resolution, it is chosen to use a grid with fully resolved boundary layer at the hub. This grid shows an average \( y^+ \) value around 1, which is recommended to predict normal gradients of velocity in the viscous sublayer in the fully resolved approach. As a remark, the average value of \( y^+ \) for the Slip Wall grid is one order of magnitude lower because \( y^+ \) is zero for walls with free-slip boundary condition.

<table>
<thead>
<tr>
<th>Grid kind</th>
<th>Nr Number of cells</th>
<th>Average ( y^+ )</th>
<th>( K_{TD} )</th>
<th>( F_{RD} )</th>
<th>CPU time[h:mm:ss]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Resolved B.L.</td>
<td>3 568 320</td>
<td>1.22E+01</td>
<td>5.91E-02</td>
<td>-3.34E-01</td>
<td>21:20:53</td>
</tr>
<tr>
<td>Slip Wall</td>
<td>2 877 120</td>
<td>6.78E-02</td>
<td>5.94E-02</td>
<td>-3.35E-01</td>
<td>20:54:57</td>
</tr>
<tr>
<td>Wall Function</td>
<td>3 043 008</td>
<td>6.32E+01</td>
<td>5.92E-02</td>
<td>-3.34E-01</td>
<td>19:32:08</td>
</tr>
</tbody>
</table>

Table 5.4: Grid details, duct loading and CPU time for the RANS-BEM computation with 20 coupling steps and three grids, with different boundary layer resolution at the hub. Computations are run on 32 cores, with 2.4GHz per core.

5.3.2. Grid refinement
With the chosen settings for the refinement of the hub boundary layer and the number of recursive steps, the grid refinement study is performed for an advance ratio of \( J=0.5 \). Six geometrically similar similar grids are given, for which the discretization error and uncertainty are evaluated with reference to both the duct thrust and duct radial force. The calculation of the numerical uncertainty is based on the method of Eça and Hoekstra[32][31] while the discretization error is computed from the extrapolated solution \( \phi_0 \):

\[
\epsilon_i = 100 \frac{\phi_i - \phi_0}{\phi_0}.
\]

The grid refinement study is executed with 1 RANS-BEM coupling step because this ensures that the same propeller force from Procal is applied to all the grids. Figure 5.4 shows the power fit with the convergence index for \( K_{TD} \) and \( F_{RD} \), while table 5.5 lists the discretization errors, uncertainties and computational time. The coarsest grid is expected to provide results out of the asymptotic range therefore it is discarded in the calculation of the fit. For these grids the convergence is monotonic but the Richardson extrapolation results in a rather unsatisfactory prediction of the convergence index, with a value of either \( p<0.5 \), or impossible to determine (as shown in the legend of the figures where \( p=\{^*1,2\} \) appears). In such cases, a large safety factor of \( F_s=3 \) is used for the uncertainty estimation[31], hence the large value of \( U_\phi \). Nevertheless, the discretization error remains below 4\% and 3.5\% for \( K_{TD} \) and \( F_{RD} \) respectively for all the data in the fit (grids 1 to 5).

<table>
<thead>
<tr>
<th>Grid Nr</th>
<th>Nr. cells</th>
<th>Rel. step size[-]</th>
<th>Error in ( K_{TD} )</th>
<th>Uncertainty ( K_{TD} )</th>
<th>Error in ( F_{RD} )</th>
<th>Uncertainty ( F_{RD} )</th>
<th>CPU time[h:mm:ss]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>16 046 000</td>
<td>1.00</td>
<td>2.38</td>
<td>7.24</td>
<td>1.69</td>
<td>5.28</td>
<td>6:28:01*</td>
</tr>
<tr>
<td>2</td>
<td>8 193 024</td>
<td>1.25</td>
<td>2.94</td>
<td>8.50</td>
<td>2.19</td>
<td>6.35</td>
<td>10:23:43</td>
</tr>
<tr>
<td>3</td>
<td>3 568 320</td>
<td>1.65</td>
<td>3.41</td>
<td>9.97</td>
<td>2.62</td>
<td>7.75</td>
<td>5:32:32</td>
</tr>
<tr>
<td>4</td>
<td>2 028 000</td>
<td>1.98</td>
<td>3.61</td>
<td>10.95</td>
<td>2.96</td>
<td>8.89</td>
<td>3:26:40</td>
</tr>
<tr>
<td>5</td>
<td>1 009 920</td>
<td>2.51</td>
<td>3.99</td>
<td>11.52</td>
<td>3.50</td>
<td>10.20</td>
<td>2:33:37</td>
</tr>
<tr>
<td>6</td>
<td>438 048</td>
<td>3.32</td>
<td>*</td>
<td>*</td>
<td>*</td>
<td>*</td>
<td>1:43:00</td>
</tr>
</tbody>
</table>

Table 5.5: Grid refinement for the RANS-BEM computations of the Ka4-70 propeller in 19A duct. Computations are run on 32 cores (except the finest grid on *128 cores) with 2.4GHz per core; simulations with 1 RANS-BEM coupling step. * not considered in the fit, hence the uncertainty is not computed.
5.3. Grid selection

For the present study, grid 3 is chosen because it provides an acceptable discretization error (3.41% in the duct thrust and 2.62% in the radial force) combined with a feasible computational cost. The uncertainty is large: 9.97% in the duct thrust and 7.75% in the radial force, but the use of a large safety factor for determining the uncertainty must be taken into account. Table 5.6 gives the details of the grid chosen for further computations and figure 5.5 shows an angular section of the grid, with the dimensions of the domain and the boundary conditions used.
5.4. Effect of hub diameter

For the chosen grid and settings, computations are run for advance ratios from 0.2 to 0.8. The results are compared with the case of smaller hub geometry \((D_{\text{HUB}}/D = 0.167)\) and with the experimental data. Open water diagrams in tabular form are reported in the Appendix. As concerns the radial force on the duct, the values from the full RANS computations are used for validation. Figure 5.6 shows the open water diagram and the plot of the radial force. Furthermore, tables 5.8 and 5.7 list the relative change with respect to the experimental data. The effect of a different hub is visible mainly at large advance ratios, with the radial force and the open water efficiency differing the most between the 2 hub geometries. The radial force is overpredicted in absolute value by the RANS-BEM computations for \(J > 0.6\), but \(K_T\) well matches with the experiments (large values of \(\%\Delta K_T\) occur at \(J = 0.6, 0.7\) because the experimental values are themselves very small). The difference in the radial force are related to the lack of the tip leakage/tip vortex in the RANS-BEM simulations. Focusing on \(J=0.5\), the results for \(D_{\text{HUB}}/D = 0.204\) are closer to the experimental values, as expected because that is the geometry used in the tests. \(K_T\) is overpredicted of 3.58% while \(K_T\) differs of 7.59%. Since the torque is also slightly overpredicted (2.24%), the open water efficiency shows a good match with a 2.15% error to the experimental result. Overall, the results for \(D_{\text{HUB}}/D = 0.204\) are closer to the experimental data than the ones for \(D_{\text{HUB}}/D = 0.167\), even though the differences are evident only at the larger Js. To conclude, the results for \(D_{\text{HUB}}/D = 0.204\) are considered in the remaining of the chapter for the comparison with Procal results as well as in the pressure distributions and contour plots.

<table>
<thead>
<tr>
<th>(J)</th>
<th>(%\Delta K_T) (%)</th>
<th>(%\Delta K_T) (%)</th>
<th>(%\Delta 10 K_Q) (%)</th>
<th>(%\Delta \eta_0) (%)</th>
<th>(%\Delta K_T) (%)</th>
<th>(%\Delta K_T) (%)</th>
<th>(%\Delta 10 K_Q) (%)</th>
<th>(%\Delta \eta_0) (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.20</td>
<td>7.55</td>
<td>5.15</td>
<td>5.12</td>
<td>1.37</td>
<td>0.20</td>
<td>5.56</td>
<td>3.78</td>
<td>3.59</td>
</tr>
<tr>
<td>0.30</td>
<td>7.58</td>
<td>5.87</td>
<td>5.38</td>
<td>1.42</td>
<td>0.30</td>
<td>5.45</td>
<td>3.71</td>
<td>3.74</td>
</tr>
<tr>
<td>0.40</td>
<td>7.03</td>
<td>6.81</td>
<td>4.97</td>
<td>1.98</td>
<td>0.40</td>
<td>4.66</td>
<td>3.58</td>
<td>3.15</td>
</tr>
<tr>
<td>0.50</td>
<td>6.56</td>
<td>12.40</td>
<td>4.42</td>
<td>3.25</td>
<td>0.50</td>
<td>3.58</td>
<td>7.59</td>
<td>2.24</td>
</tr>
<tr>
<td>0.60</td>
<td>7.24</td>
<td>37.89</td>
<td>4.55</td>
<td>6.03</td>
<td>0.60</td>
<td>4.18</td>
<td>21.62</td>
<td>2.36</td>
</tr>
<tr>
<td>0.70</td>
<td>10.24</td>
<td>63.00</td>
<td>6.02</td>
<td>9.98</td>
<td>0.70</td>
<td>0.98</td>
<td>13.27</td>
<td>0.49</td>
</tr>
<tr>
<td>0.80</td>
<td>4.12</td>
<td>10.84</td>
<td>0.15</td>
<td>22.26</td>
<td>0.80</td>
<td>12.95</td>
<td>0.79</td>
<td>10.60</td>
</tr>
</tbody>
</table>

Table 5.6: Geometry, topology and settings for the chosen grid for the RANS-BEM computations of the Ka4-70 propeller in 19A duct.


Table 5.7: Relative change wrt experimental. RANS-BEM, small hub \(D_{\text{HUB}}/D = 0.167\). (In the model tests \(D_{\text{HUB}}/D = 0.204\).)

Table 5.8: Relative change wrt experimental. RANS-BEM, large hub \(D_{\text{HUB}}/D = 0.204\). (In the model tests \(D_{\text{HUB}}/D = 0.204\).)
5.5. BEM and RANS-BEM solutions

The solution of the RANS-BEM approach is here compared with the Procal solution. Figures 5.7 show the complete open water diagram and a zoom-in on the propeller and duct thrust. The open water efficiency is mispredicted in the Procal solution and largely differ with both the RANS-BEM and the experimental data at J>0.5 because $K_{TD}$ is overestimated. This is due to flow separation at the outer side of the duct which is not modeled in the potential flow. In terms of relative change with the experimental data (as in tables 5.9 and 5.10), there is an error of over 27% in $K_{TD}$ for Procal at J=0.5, which drops at 7.59% for the RANS-BEM solution. Also at larger loading (J=0.2) there is an improvement in the coupled solution, where the error is 3.78% compared to 11.14% for the BEM reference computation. In addition, close to the design condition J=0.55, both $K_{TP}$ and $K_Q$ are better predicted with the RANS-BEM approach with errors around 4% for the thrust and 2% for the torque which then leads to a good prediction of open water efficiency, where there is a relative change in $\eta_0$ of 2.15% at J=0.5 and 3.79% at J=0.6. Overall, the RANS-BEM coupling provides more accurate results for the duct thrust throughout the whole open water range and better prediction of propeller thrust and torque close to design condition, which results in a good match for the open water efficiency with the experimental data.
5.6. Pressure distributions on the propeller and on the duct

For the Procal solution and the ReFRESCO-Procal solution the pressure distribution on the duct and at the blade are looked at. Figure 5.8 shows the mean pressure distribution on the duct for two advance ratios: J=0.5 and J=0.8. The BEM Reference solution refers to the Procal solution as discussed in the previous chapter. The item BEM Coupled stands for the Procal solution obtained at the last step of the RANS-BEM coupling. Both of them requires the averaging of the pressure distribution over the blade angle to obtain the mean pressure, and the duct extends up to $x/c = 1.08$ because of the longer duct geometry used in Procal. The RANS-coupled result is the axisymmetric pressure distribution from ReFRESCO in the RANS-BEM coupling and the Full RANS is the mean pressure computed from the validation data. In view of the coupling procedure, the duct radial force for the BEM Coupled and the RANS-coupled is equal. At J=0.5 the distributions matches well at the duct outer side, whereas at the inner surface there is a slight underestimation up to $x/c = 0.6$ in Procal when compared to the RANS-BEM, which then balances with an overestimation more downstream. When comparing the RANS with body force and the full RANS solution, the latter shows a larger pressure between $x/c = 0.1$ and $x/c = 0.4$, but in correspondence of the propeller axial position ($x/c$ from 0.4 to 0.6) the pressure is lower. This is due to the effect of the tip leakage vortex forming in the gap between the blade and the duct, where the pressure drops and therefore the mean value is affected. These two effects balances each other such that both the duct thrust and the radial force show a good match (figure 5.6 at J=0.5). For the advance ratio J=0.8 the larger differences occur at the duct outer surface: as presented in chapter 4, Procal mispredicts the pressure because the flow remains attached in the potential flow solution; the RANS-BEM solution is almost constant around a value of $CPN = -0.3$ but the full RANS pressure is lower up to $x/c=0.6$ and larger towards the trailing edge. This difference in pressure relates to the different pattern of the recirculating flow. As concerns the duct inner side, the RANS-coupled constantly underpredicts the pressure; this results in a larger (in absolute value) radial force. In addition, it is noticeable that the pressure at the duct trailing edge is underpredicted by the RANS-coupled, with the full RANS value closer to the Procal solutions.

![Figure 5.8: Duct pressure distribution at J=0.5. Mean BEM Reference solution, mean BEM coupled solution and axisymmetric RANS-coupled distribution compared to a full RANS solution.](image)
The pressure distributions on the blade are also compared, between the Procal reference, Procal coupled and the full RANS solution. Figures 5.9 show the chordwise distribution at radial position \( r/R = 0.5 \) and \( r/R = 0.9 \) for \( J=0.5 \). As for the former, the distribution for Procal shows a good prediction up to 80\% the chord length, with the coupled solution matching very well the full RANS distribution. The differences towards the trailing edge are related to the pressure recovery in the potential flow code and the modeling of the propeller trailing edge in the full RANS and the BEM. At the larger radius \( r/R = 0.9 \), there is a larger mismatch because the pressure in the full RANS solution is influenced by the tip leakage vortex.

5.7. Contour plots

Figures 5.10 show the contours of the body force in X direction and the pressure coefficient. The propeller swept volume extends from \( x/R \approx -0.15 \) to \( x/R \approx 0.15 \), with the body force being negative over the whole volume, except a region close to the hub corresponding to the blade leading edge at small radii. At the mid-chord positions the force is maximum (in absolute value). The streamlines show that the flow remains attached at the duct outer surface at \( J=0.5 \). Also, no separation occurs at the duct inner surface, which was observed just aft of the blade location in previous studies[12] when the correction of additional body forces in the gap was not used. The pressure coefficient \( CPN \) shows the stagnation point at the duct nose and the drop in pressure at the inner surface with the minimum at \( x/R = 0.35 \). Flow recirculation occurs at the duct trailing edge at \( J=0.5 \).

Furthermore, figures 5.11 illustrate the flow separation at the duct outer surface for \( J=0.8 \). The contour plots give the dimensionless vorticity \( \omega' \) (defined in chapter 2) and the dimensionless axial velocity \( V_x/V_0 \). The vorticity is large at the duct inner side where the boundary layer builds up, as well as in the shear layer.
close to the leading edge where separation occurs. The contour of axial velocity (together with the stream-
lines) clearly show the reverse flow in the recirculating region.

![Figure 5.11: J=0.8. Streamlines and contour plots of the dimensionless vorticity $\omega^\prime$ and the axial velocity. Separation at the duct outer surface is observed at large advance ratios.](image)

### 5.8. Summary

From the RANS-BEM computations of the Ka4-70 propeller in 19A duct, it is concluded that the interpolation settings for the body force distribution largely influence the duct loading. The volume intersection method requires a sufficient number of refinement steps in the recursive algorithm. Concerning the two geometries with different hub diameters, it is seen that the difference does not affect significantly the results, except at the larger advance ratios. The grid refinement for the RANS grid led to acceptable errors and numerical uncertainties, even though a poor convergence behavior results in large safety factors. The RANS-BEM approach improves the prediction of duct thrust and radial force in comparison to the pure Procal simulations, with a good match of the duct loading also at large advance ratios when the flow separates at the outer surface. To continue with, the comparison of the duct mean pressure with the full RANS solution at $J=0.5$ reveals a balancing effect, which leads to a correct integral value of duct thrust and radial force. Differently, at $J=0.8$ the pressure distribution differs at both duct inner and outer surface, therefore the radial force differs (but the duct thrust is still well predicted). Finally, the chordwise pressure distribution on the blade shows the difference between the Procal and ReFRESCO computations: the modeling of the blade trailing edge which affects the pressure at all radial location and the lack of tip leakage vortices in Procal which results in the observed mismatch in pressures towards the blade tip.
This chapter presents the results for the boundary element method calculation of the Ka4-70 propeller in duct 37. A grid refinement study for the duct surface is carried out (section 6.2) at first. With the chosen grid, the position of the modified trailing edge is studied, using the pressure-based iterative scheme: the outcome are presented in section 6.3. Moreover, section 6.4 shows the computed pressure distributions and contours on the blade and duct surfaces and finally section 6.5 presents a further investigation on the effect of a modified duct trailing edge geometry by a 2D integration of the mean pressure on the duct.

6.1. Introduction

The duct 37 is a difficult test case for the boundary element method because major changes are necessary to adapt the profile for the potential flow calculations with Procal, with the position of the duct trailing edge that plays a major role in the predicted performances. The pressure-equality scheme presented in chapter 4 is applied to this geometry as well, and the solution is compared with the experimental open water diagrams. The limitations of the potential flow analysis are investigated (sections 6.4 and 6.5) by a comparison of the pressure distributions with the full RANS and a 2D analysis of the forces on the duct. It is useful to bear in mind that the validation material for this geometry is limited because only open water tests were carried out and the full RANS results are only partially available, which poses a limit to the some conclusions.

6.2. Grid refinement study

Due to the lack of data on the grid sensitivity for the duct 37, a refinement study is carried out. The propeller is paneled in the same way as the previous 19A duct test case. The grid refinement is first of all executed for the panels in axial direction on the duct (i.e. from the leading edge to the trailing edge) and secondly for the panels in circumferential direction. The study is done for an advance ratio at J=0.5. Table 6.1 gives the details of the paneling on the duct, where the number of panels between the blades is kept constant. Figures 6.1 give the computed numerical uncertainty for the duct thrust and the radial force. The two coarser grids are out of the asymptotic range, therefore they are not taken into account for the extrapolation of the estimated exact solution. The duct thrust shows a convergence of second order, while the radial force shows first order convergence. The uncertainty for the 6 grids is maximum 7.2% for the thrust and 5.1% for the radial force. Table 6.2 lists also the error calculated from the extrapolated exact solution. The error is below 5% and 2% for the thrust and radial force respectively. In order to keep the error for $K_{TD}$ also around 2%, grid 3 is chosen. With this configurations, the uncertainty becomes 6.5% for $K_{TD}$ and 1.68% for $F_{RD}$.
6. Procal computations of the Ka4-70 propeller in 37 duct

<table>
<thead>
<tr>
<th>Grid number</th>
<th>Panels  (inner:outer)</th>
<th>LE</th>
<th>Spacing</th>
<th>TE</th>
<th>Between blades</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4250(120x25:50x25)</td>
<td>0.003</td>
<td>0.006</td>
<td>0.011</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>3875(110x25:45x25)</td>
<td>0.0035</td>
<td>0.007</td>
<td>0.011</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3500(100x25:40x25)</td>
<td>0.004</td>
<td>0.008</td>
<td>0.011</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>3125(90x25:35x25)</td>
<td>0.006</td>
<td>0.009</td>
<td>0.011</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>2750(80x25:30x25)</td>
<td>0.007</td>
<td>0.010</td>
<td>0.011</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>2375(70x25:25x25)</td>
<td>0.008</td>
<td>0.011</td>
<td>0.011</td>
<td></td>
</tr>
</tbody>
</table>

Table 6.1: Settings for the grid refinement study for the paneling on the duct in axial direction.

<table>
<thead>
<tr>
<th>Grid Nr</th>
<th>Rel. step size[-]</th>
<th>Error in $K_{TD}$</th>
<th>Uncertainty $K_{TD}$</th>
<th>Error in $F_{RD}$</th>
<th>Uncertainty $F_{RD}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.00</td>
<td>1.71</td>
<td>5.3</td>
<td>1.48</td>
<td>4.4</td>
</tr>
<tr>
<td>2</td>
<td>1.05</td>
<td>1.89</td>
<td>5.8</td>
<td>1.51</td>
<td>4.7</td>
</tr>
<tr>
<td>3</td>
<td>1.10</td>
<td>2.01</td>
<td>6.5</td>
<td>1.68</td>
<td>4.9</td>
</tr>
<tr>
<td>4</td>
<td>1.66</td>
<td>2.35</td>
<td>7.2</td>
<td>1.70</td>
<td>5.1</td>
</tr>
<tr>
<td>5</td>
<td>1.24</td>
<td>*</td>
<td>*</td>
<td>*</td>
<td>*</td>
</tr>
<tr>
<td>6</td>
<td>1.34</td>
<td>*</td>
<td>*</td>
<td>*</td>
<td>*</td>
</tr>
</tbody>
</table>

Table 6.2: Numerical error and uncertainty for the grid refinement in axial direction. CPU time is between 25 and 40 minutes, without a clear dependence on grid refinement. * not in the fit.

Figure 6.1: Grid convergence study for the Ka4-70 propeller in 37 duct; the paneling of the duct surface is refined in axial direction. J=0.5.

Secondly, the paneling of the duct is refined in circumferential direction, keeping constant the density in axial direction. Table 6.3 and figures 6.2 provide the details of the Procal settings and the results of the grid convergence index study. For the duct radial force, the observed order of convergence is $p = 2$ while for the duct force the order is either below 0.5 or not possible to determine. This results in larger safety factors (section 2.6 from the theoretical background) in the computed uncertainty. The two coarsest grids are outside of the asymptotic range; the solution for the coarsest grid 3.6 is also not converged, so the computed values are significantly different than for the finer grids. Table 6.4 gives the error and the numerical uncertainties for all the grids. The numerical uncertainty does not fall below 5% for the duct thrust, while it remains between 1% and 3% for the radial force. In this case, it is decided to retain grid 3.3 for the Procal computations. It provides an acceptable error of 2.28% for the duct thrust and a negligible 0.6% for the radial force. The uncertainty is in line with the computed values for the refinement in the axial direction, i.e. 6.7%.

As a sum up, tables 6.5 and 6.6 give the geometrical details, the operating condition and the paneling specification for the final grid used for the BEM computations.
Table 6.3: Settings for the grid refinement study for the paneling on the duct in angular direction.

<table>
<thead>
<tr>
<th>Grid number</th>
<th>Panels (inner:outer)</th>
<th>Spacing LE</th>
<th>Spacing TE</th>
<th>Between blades</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>(100x40:40x40)</td>
<td>0.004</td>
<td>0.008</td>
<td>0.004</td>
</tr>
<tr>
<td>3.2</td>
<td>(100x35:40x35)</td>
<td>0.004</td>
<td>0.008</td>
<td>0.007</td>
</tr>
<tr>
<td>3.3</td>
<td>(100x25:40x25)</td>
<td>0.004</td>
<td>0.008</td>
<td>0.011</td>
</tr>
<tr>
<td>3.4</td>
<td>(100x20:40x20)</td>
<td>0.004</td>
<td>0.008</td>
<td>0.02</td>
</tr>
<tr>
<td>3.5</td>
<td>(100x15:40x15)</td>
<td>0.004</td>
<td>0.008</td>
<td>0.04</td>
</tr>
<tr>
<td>3.6</td>
<td>(100x10:40x10)</td>
<td>0.004</td>
<td>0.008</td>
<td>0.06</td>
</tr>
</tbody>
</table>

Table 6.4: Numerical error and uncertainty for the grid refinement in angular direction. ⋆ not in the fit.

Grid Nr | Rel. step size | Error in $K_{TD}$ | Uncertainty $K_{TD}$ | Error in $F_{RD}$ | Uncertainty $F_{RD}$
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>1.00</td>
<td>1.92</td>
<td>5.7</td>
<td>0.41</td>
<td>1.1</td>
</tr>
<tr>
<td>3.2</td>
<td>1.17</td>
<td>2.09</td>
<td>6.2</td>
<td>0.075</td>
<td>1.5</td>
</tr>
<tr>
<td>3.3</td>
<td>1.42</td>
<td>2.28</td>
<td>6.7</td>
<td>0.60</td>
<td>1.8</td>
</tr>
<tr>
<td>3.4</td>
<td>1.79</td>
<td>2.38</td>
<td>7.0</td>
<td>0.79</td>
<td>2.6</td>
</tr>
<tr>
<td>3.5</td>
<td>2.43</td>
<td>⋆</td>
<td>⋆</td>
<td>⋆</td>
<td>⋆</td>
</tr>
<tr>
<td>3.6</td>
<td>3.78</td>
<td>⋆</td>
<td>⋆</td>
<td>⋆</td>
<td>⋆</td>
</tr>
</tbody>
</table>

Table 6.5: Geometry and wake model settings for the Procal computations of Ka4-70 in 37 duct.

<table>
<thead>
<tr>
<th>Duct</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades Z</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>Propeller model</td>
<td>5979</td>
<td></td>
</tr>
<tr>
<td>Diameter D</td>
<td>0.24m</td>
<td></td>
</tr>
<tr>
<td>Hub diameter $D_{HUB}/D$</td>
<td>0.167</td>
<td></td>
</tr>
<tr>
<td>rotation rate $n$</td>
<td>15 rps</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Wake model</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades Z</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>Propeller model</td>
<td>5979</td>
<td></td>
</tr>
<tr>
<td>Diameter D</td>
<td>0.24m</td>
<td></td>
</tr>
<tr>
<td>Hub diameter $D_{HUB}/D$</td>
<td>0.167</td>
<td></td>
</tr>
<tr>
<td>rotation rate $n$</td>
<td>15 rps</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Spacings</th>
<th>Propeller Spacings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blade panels</td>
<td>LE</td>
</tr>
<tr>
<td>1500 (2x30x25)</td>
<td>0.001</td>
</tr>
<tr>
<td>Duct Spacings</td>
<td>Panels (inner:outer)</td>
</tr>
<tr>
<td>3500(100x25:40x25)</td>
<td>0.004</td>
</tr>
<tr>
<td>Hub Spacings</td>
<td>Hub panels</td>
</tr>
<tr>
<td>800(80x10)</td>
<td>0.02</td>
</tr>
</tbody>
</table>

Table 6.6: Blade, duct and hub meshing with values for the spacings.
6.3. Trailing edge geometry

The boundary element computation requires a sharp trailing edge. The 37 duct has a blunt trailing edge. While the outer duct surface is almost a straight line, the inner surface has a large curvature. This means that the modification to provide a sharp trailing edge are more radical than for the duct 19A, as visible in figure 6.3. This makes the computation sensitive to the choice of modified duct geometry. It is proved in [34] that, similarly to the 19A tests case, the axial position of the trailing edge has not much influence on the loading; however, the radial position significantly affects the propeller thrust. The iterative scheme described in chapter 4 is applied. The scheme is based on a secant method and the criterion is to match the pressure at two locations, on the inner side and on the outer side of the duct. The governing equation is here recalled:

$$\Delta C_{pn} = C_{pn, outer} - C_{pn, inner} = f \left( \frac{y_t}{L} \right),$$

(6.1)

and the radial position of the trailing edge $y_t/L$ is computed iteratively with a secant method such that $\Delta C_{pn}$ tends to zero. For the duct 37 the outer surface is largely modified and it is much more curved than the original geometry. As a result, the BEM predicts a pressure recovery at the duct outer surface which is not physical. When the pressures is matched at two points on the outer and inner surface both located at $x/R = 1$ (i.e. in correspondence on the original trailing edge), this pressure will be too large, leading to an overprediction of the propeller loading. Therefore, the same control points as for duct 19A, cannot be used for duct 37, as illustrated in figure 6.3. For the outer surface of the nozzle the location for equal pressure must be selected at a more upstream location, where the modification of the geometry has less influence. For this study, a position at $x/R = 0.5$ is chosen for the duct outer surface, equal for all the open water range considered. At the inner surface $x/R = 1$ is used. Furthermore, the tolerance in the iterative scheme is retained to $1E-04$, this being the absolute difference of the pressure coefficients. This tolerance is found to be rather small, with the scheme that reaches sometimes the maximum number of iterations prescribed (30). Since the pressure difference quickly drops by 2 order of magnitudes in the first 5 to 10 iterations, the tolerance could be increased to $1E-03$ with a minor loss in accuracy. Concerning the initial condition, computations are started with initial conditions from $y/L = 0.02$ to $y/L = 0.10$. For the larger values, the BEM code is not stable, therefore difficulties in convergence are sometimes observed; initial conditions towards the extension of the inner surface are therefore preferable.

<table>
<thead>
<tr>
<th>Advance ratio J</th>
<th>Radial position T.E. $\frac{y_t}{L}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2</td>
<td>0.049*</td>
</tr>
<tr>
<td>0.3</td>
<td>0.043*</td>
</tr>
<tr>
<td>0.4</td>
<td>0.045</td>
</tr>
<tr>
<td>0.5</td>
<td>0.049</td>
</tr>
<tr>
<td>0.6</td>
<td>0.055</td>
</tr>
<tr>
<td>0.7</td>
<td>0.063</td>
</tr>
<tr>
<td>0.8</td>
<td>0.075</td>
</tr>
<tr>
<td>Formerly[34] (all J)</td>
<td>0.05</td>
</tr>
</tbody>
</table>

Table 6.7: Radial position of the trailing edge obtained with the iterative scheme for the pressure difference. * not converged.

Figure 6.4: Geometries for the initial $y/L_{TE}$ and converged geometry for J=0.5.
6.3. Trailing edge geometry

Figure 6.4 shows the trailing edge geometry for two initial conditions and for the converged solution at $J=0.5$. Table 6.7 lists the obtained radial position of the trailing edge. As for the 19A duct, the computed position is larger for larger advance ratios. A reference value is suggested in [34] where Procal computations with duct 37 are executed, but with a different propeller from the D-series[23]. The effect of the radial position of the trailing edge is visible in figures 6.5 where the spanwise thrust distribution on the propeller and the chordwise mean pressure distribution on the duct are presented for the converged solution and the initial conditions at $J=0.5$. The force on the propeller is too small for the larger $y/L_{TE}$, which relates to a large force on the duct, as the small pressure on the inner side of the duct for $y/L_{TE}=0.09,0.10$ testifies. For completeness, the plot of the mean pressure distribution also shows two red dots which correspond to the points where the pressure is made equal at the nozzle sides. As a conclusion, the advantage of the iterative scheme developed in this work is that there is no need for a detailed sensitivity study or observations of the flow behavior at the duct trailing edge (from CFD) in order to have a proper setting for the radial position. Nonetheless, for the 37 duct some insight is required because the pressure-equality condition is not satisfactory if applied to a point on the duct outer side where a considerable pressure recovery occurs.

In addition to the iteratively computed trailing edge, calculations were executed for a fixed trailing edge throughout the open water range. This is tuned to a value of $y/L=0.05$ by matching the propeller thrust with the experimental value at $J=0.5$. This solution is referred to as base solution and the modified duct geometry is shown in figure 6.7. Because a different trailing edge location for different $J$s will result in different panel files for the different loading condition, it is convenient to use a single trailing edge location for comparison with the hybrid scheme, hence the base computation is used as reference. The open water diagram 6.6 shows
the solutions for the iterative trailing edge calculations and the fixed trailing edge base solution while table 6.8 and 6.9 give the relative change with respect to the experimental values. The base and iterative solution provide the same forces at the design condition but the difference between the two gets larger for very small and very large advance ratios. At high loading condition, the base solution overpredicts the torque and the iterative solution underpredict the thrust. Viceversa, at large J the torque is too large for the base solution and the propeller thrust is underpredicted in the iterative solution. The duct thrust shows small difference between the two solution and finally the open water efficiency is constantly overpredicted, with the iterative solution providing a better prediction at light loading. Looking at the errors compared to the experimental solution, the duct thrust shows the largest discrepancies (getting meaningless large when $K_{TD}$ is itself very small in absolute sense). Finally, the open water efficiency for both solutions is overpredicted of approximately 10% in design and heavy loading conditions, but the overprediction is much larger for light loading (>20% at J=0.7). In the following section results are presented for the pressure distribution at J=0.5. It is chosen to show here the results for the base solution, nevertheless the discrepancy with the iterative trailing edge solution at design condition is small.

### 6.4. Pressure on the duct and on the blade surfaces

The pressure distributions for the Procal solution are compared with the full RANS solution[14] for the blade surface and the duct at J=0.5. The contour plots in figure 6.8 show the two sides of the blade for the potential flow and viscous calculations. As for the Ka4-70 in 19A duct, the larger differences are seen toward the leading edge at the pressure side and at the blade tip. The RANS computations shows a region of low pressure at the leading edge at lower radii at the pressure side. The pressure there is expected to be influenced by the mesh density, but information is not available in this sense for the full RANS.

In addition, there is an elongated region of low pressure at the suction side at the blade tip which is due to the development of the tip leakage vortex, which is not present in the BEM. Also, the RANS solution shows a region of lower pressure at the leading edge for $r/R > 0.8$. This is due to a large pressure drop just after the stagnation point, which becomes wider for the large radial position where the tangential velocity is larger. X-Y plots of the chordwise pressure distribution on the blade are discussed in chapter 7 together with the hybrid RANS-BEM results.

Moreover, the pressure distributions on the duct surface are compared in figure 6.9. Plots are given for two angular position: 0° and 40°. The full RANS distribution at 40° is rather oscillatory, which is probably due to interpolation issues. At 0°, there is a jump in pressure due to the pressure difference between the blade sides. Procal underpredicts the pressure at the inner surface, with a visible underprediction at 0.15 < $x/c$ < 0.5. The full RANS solution shows a pressure peak at $x/c = 0.75$ which is due to the tip leakage vortex in the wake. Nevertheless, it is noticeable that the pressure peak is rather wider than in the full RANS for the 19A duct. This is likely due to the effect of a coarser mesh. Indeed, it is experienced that a coarse mesh results in weaker vortices with a larger viscous core radius. At the outer duct surface, Procal shows a pressure recovery at $x/c > 0.8$ which is not present in the viscous solution. Regarding the distribution at 40°, Procal underpredicts the pressure at the inner surface and the pressure is higher at the trailing edge location. The peak due to the tip leakage vortex is present at $x/c = 0.65$ for the full RANS solution. Overall, it is concluded that the two main mispredictions in the BEM are the underprediction upstream of the propeller and the pressure recovery towards the trailing edge. Considering that the inner surface in the modified geometry is almost straight, the underprediction at the inner side does not influence the duct thrust, instead the pressure recovery at the outer side (where the surface is curved) leads to the overpredicted $K_{TD}$. Since the prediction of propeller thrust and torque are accurate at J=0.5 for the BEM, it is believed that the radial force (which is the major

<table>
<thead>
<tr>
<th>J</th>
<th>$\Delta K_{TP}$ (%)</th>
<th>$\Delta K_{TD}$ (%)</th>
<th>$\Delta 10 K_Q$ (%)</th>
<th>$\Delta \eta_B$ (%)</th>
<th>J</th>
<th>$\Delta K_{TP}$ (%)</th>
<th>$\Delta K_{TD}$ (%)</th>
<th>$\Delta 10 K_Q$ (%)</th>
<th>$\Delta \eta_B$ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.20</td>
<td>0.52</td>
<td>15.18</td>
<td>1.57</td>
<td>8.46</td>
<td>0.20</td>
<td>0.39</td>
<td>15.18</td>
<td>2.35</td>
<td>8.78</td>
</tr>
<tr>
<td>0.30</td>
<td>1.11</td>
<td>18.04</td>
<td>2.71</td>
<td>8.80</td>
<td>0.30</td>
<td>4.89</td>
<td>16.47</td>
<td>2.06</td>
<td>6.86</td>
</tr>
<tr>
<td>0.40</td>
<td>1.57</td>
<td>21.66</td>
<td>2.76</td>
<td>8.70</td>
<td>0.40</td>
<td>3.10</td>
<td>20.76</td>
<td>0.89</td>
<td>7.67</td>
</tr>
<tr>
<td>0.50</td>
<td>0.94</td>
<td>29.82</td>
<td>2.18</td>
<td>9.18</td>
<td>0.50</td>
<td>0.12</td>
<td>29.66</td>
<td>1.56</td>
<td>9.07</td>
</tr>
<tr>
<td>0.60</td>
<td>3.43</td>
<td>61.87</td>
<td>0.81</td>
<td>12.23</td>
<td>0.60</td>
<td>4.45</td>
<td>62.42</td>
<td>4.91</td>
<td>12.20</td>
</tr>
<tr>
<td>0.70</td>
<td>15.89</td>
<td>3631.67</td>
<td>8.19</td>
<td>27.87</td>
<td>0.70</td>
<td>10.24</td>
<td>3536.67</td>
<td>9.16</td>
<td>22.81</td>
</tr>
<tr>
<td>0.80</td>
<td>59.32</td>
<td>113.04</td>
<td>27.78</td>
<td>219.90</td>
<td>0.80</td>
<td>28.37</td>
<td>104.41</td>
<td>18.00</td>
<td>121.67</td>
</tr>
</tbody>
</table>

Table 6.8: Relative change wrt experimental. BEM, duct 37, base trailing edge position.

Table 6.9: Relative change wrt experimental. BEM, duct 37, iterative trailing edge position.
6.4. Pressure on the duct and on the blade surfaces

Figure 6.8: Blade pressure contours for the Procal reference solution and the full RANS solution[14], J=0.5.

Figure 6.9: Chordwise pressure distribution on the duct for the BEM and full RANS[14] solutions at angular position of 0° and 40°. The oscillatory distribution in the RANS is likely due to a poor interpolation.
component of the lift, hence relates to circulation) is also correctly predicted. In this case there would be a balancing effect, with the underprediction of BEM pressure at the inner side and the recovery at the trailing edge balancing each other. However, the lack of full RANS values for the radial force does not allow to draw strong conclusions on that.

6.5. Integration of pressure over the original geometry

The change in duct geometry to generate a sharp trailing edge results in major changes in the duct profile. A further task is carried out to evaluate what the effect of such change is for the duct thrust. A 2D integration of mean pressure over the duct profile is performed by a discretization of the duct sections. The analysis is carried out for the operating condition $J=0.5$ only. Figures 6.10 show the 2D paneling of the original and modified geometry. The former is discretized using the points for the standard definition of the 37 duct, whereas the latter is discretized as in the Procal solution. The outward oriented unit normal vectors are also shown, with origin at the control points at the center of the panels and a polar reference system $(X, R, \theta)$ is used. In the implementation, the inner and outer surfaces of the duct are kept separate to ease the interpolation of the pressures at the location of the control points. To ensure the normal vector to point outward, the normals are computed by:

$$
n_{\text{inner}} = s \times e_\theta$$  \hspace{1cm} \text{for the inner surface} \hspace{1cm} (6.2)$$
$$
n_{\text{outer}} = -s \times e_\theta$$  \hspace{1cm} \text{for the outer surface} \hspace{1cm} (6.3)$$

where $e_\theta$ is the unit vector for the out-of-plane angular direction $e_\theta = (0, 0, 1)$ and the vector $s$ is the tangent vector normalized with the panel length. The duct thrust is then computed by integration of the pressure:

$$K_{TD} = -2\pi \rho V_{\text{ref}}^2 D^4 \int p n_s \, ds = -2\pi \rho V_{\text{ref}}^2 D^4 \sum_{i=1}^{N_{\text{panels}}} p_i n_s \, l_i,$$  \hspace{1cm} (6.4)$$

with $l_i$ being the length of the panel, the reference velocity $V_{\text{ref}} = nD$ as in Procal, and the factor $2\pi$ to consider the whole duct.

As concerns the pressure, three distributions are considered, plotted in figure 6.11. The BEM reference distribution is the solution from Procal, whereas the Adapted distributions are generated manually by prescribing the value of $CPN$ at the trailing edge and impose a linear distribution between the trailing edge and a selected point on the inner and the outer surface. The idea is to check what is the effect of excluding the unphysical pressure recovery towards the trailing edge which occurs for the BEM. Two "artificial" distributions are created, corresponding to values of $CPN$ at the trailing edge of 0.03 and -0.03 respectively.

Figure 6.10: Section of the original and modified duct geometry and surface paneling. The unit outward-pointing normal vectors are plotted. For the paneling of the original 37 duct, the corner points are chosen according to the standard definition of the 37 duct section. For the modified geometry, the paneling results from the surface used in the Procal simulations.

For this investigation, the geometry of the original 37 duct are taken from Prove (the meshing tool for Procal) where the geometry of duct 37 is saved.
The results of the 2D integration for the three pressure distributions are summarized in table 6.10 where also the experimental value is presented and the Procal result. When the BEM reference distribution is considered, the integration over the modified profile provides a value of $K_{TD}$ very close to the Procal solution, as expected. Remarkably, also the integration over the original geometry gives a similar value, with a roughly 30% error with respect to the experimental again. This means that the change in geometry does not affect the duct thrust for the BEM pressure distribution. The reason relates to the pressure recovery at the aft part: for the modified geometry the pressure recovery at the outer side provides an excessive positive thrust component, while for the original geometry the pressure peak at the trailing edge involves a local large axial force because the normal vector has a large axial component at the original blunt trailing edge.

When the adapted pressure distributions are integrated, the discrepancy with the experimental result falls to 9.07% for the modified geometry and to 0.24% for the original geometry. The difference between the two geometries gets larger than the case of the BEM reference pressure but the errors are considerably improved. To conclude, this analysis shows that the reason why a modified geometry leads to overprediction is the pressure recovery (particularly for the outer surface) which occurs in the potential flow simulation. Future work should focus on modeling a correct pressure, starting for instance with the effect of applying the transpiration velocity model on the outer surface to change the duct effective camberline.

6.6. Summary

The Procal simulations of the Ka4-70 propeller in 37 duct show that accurate prediction of the propeller thrust and torque (<2% error) are achieved with a proper choice of the trailing edge radial position. This is possible by mean of the iterative scheme or by tuning the radial position as in the base solution. However, the duct thrust remains overpredicted and a closer look at the pressure distribution reveals a pressure recovery at the aft part of the duct. Finally, the 2D integration of the mean pressure reveals that no changes are obtained if the same pressure is integrated over the original and the modified duct geometry, so it confirms that the overprediction in $K_{TD}$ is caused by the excessive pressure towards the trailing edge predicted by Procal.
The chapter gives the results of the hybrid computations of the Ka4-70 propeller in 37 duct. A grid refinement is carried out in section 7.2 for the RANS mesh with reference to the duct thrust and radial force. The results are then compared in section 7.3 with the Procal solution; more detailed results are given in section 7.4 for the pressure distributions and in section 7.5 for the contour plots of velocity and vorticity, where also the issue of flow separation is discussed.

7.1. Introduction

The hybrid computations of the Ka4-70 in 37 duct are performed here; the settings in the RANS-BEM simulation, and the domain characteristics, are the same as in the case of the 19A duct. The coupling requires the user to set the number of coupling steps, which is here set to 20. Only for the grid refinement 1 coupling step is used. The results are analyzed in form of open water diagrams for the integral quantities (thrust, torque), pressure distributions and contour plots. The pressure distributions are particularly important because they let a comparison of the BEM, the axisymmetric RANS solution and the validation material from full RANS calculation. Nonetheless, as presented in chapter 3, the full RANS dataset was not entirely available, which limits some of the conclusions. The effect of the hub diameter is not investigated because in all data sets have a consistent diameter of $D_{HUB}/D = 0.167$ is used.

7.2. Grid refinement

A grid refinement study is carried out to determine the grid density for the RANS computations. The computational domain has the same dimensions of the grids used for the 19A geometry (figure 5.5). Also, it is assumed that the same considerations for the boundary condition at the hub surface are valid, so a mesh with a fully resolved boundary layer is adopted, which results in a minor increase in computational time when compared with a grid with wall functions. The grid refinement study is carried out for 1 RANS-BEM coupling step because in such way the propeller force interpolated from the boundary element method to the axisymmetric viscous solution is the same. The grid study is executed for an advance ratio of $J=0.5$. 
Table 7.1: Grid refinement for the RANS-BEM computations of the Ka4-70 propeller in 37 duct. Computations are run on 32 cores (except the finest grid on *128 cores) with 2.4GHz per core; simulations with 1 RANS-BEM coupling step. ⋆ Not in the fit, so the numerical uncertainty is not computed.

<table>
<thead>
<tr>
<th>Grid Nr</th>
<th>Number of cells N</th>
<th>Rel. step size [-]</th>
<th>Error in $K_{TD}$ $\epsilon_i$ [%]</th>
<th>Uncertainty $K_{TD}$ $U_{\phi}$ [%]</th>
<th>Error in $F_{RD}$ $\epsilon_i$ [%]</th>
<th>Uncertainty $F_{RD}$ $U_{\phi}$ [%]</th>
<th>CPU time [h:mm:ss]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>16 046 400</td>
<td>1.00</td>
<td>2.50</td>
<td>7.7</td>
<td>3.10</td>
<td>9.5</td>
<td>15:01:54*</td>
</tr>
<tr>
<td>2</td>
<td>8 355 840</td>
<td>1.25</td>
<td>3.05</td>
<td>8.8</td>
<td>3.73</td>
<td>10.8</td>
<td>9:50:54</td>
</tr>
<tr>
<td>3</td>
<td>3 632 256</td>
<td>1.65</td>
<td>3.28</td>
<td>9.7</td>
<td>4.09</td>
<td>11.9</td>
<td>4:26:46</td>
</tr>
<tr>
<td>4</td>
<td>2 047 200</td>
<td>1.99</td>
<td>3.37</td>
<td>10.0</td>
<td>4.12</td>
<td>12.3</td>
<td>3:10:35</td>
</tr>
<tr>
<td>5</td>
<td>1 038 336</td>
<td>2.51</td>
<td>3.21</td>
<td>9.4</td>
<td>3.91</td>
<td>11.3</td>
<td>2:19:32</td>
</tr>
<tr>
<td>6</td>
<td>454 032</td>
<td>3.30</td>
<td>*</td>
<td>*</td>
<td>*</td>
<td>*</td>
<td>2:14:51</td>
</tr>
</tbody>
</table>

Table 7.2: Geometry, topology and settings for the chosen grid for the RANS-BEM computations of the Ka4-70 propeller in 37 duct. *See reference[15] for the settings of the control file for ReFRESCO and [7] for the theoretical background. The complete control file is given in the Appendix.

<table>
<thead>
<tr>
<th>Grid characteristics</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Hub geometry</td>
<td>Finite hub upstream, infinite hub downstream</td>
</tr>
<tr>
<td>Hub boundary layer resolution</td>
<td>No-slip condition without wall functions</td>
</tr>
<tr>
<td>$D_{HUB}/D$</td>
<td>0.167</td>
</tr>
<tr>
<td>Number of cells</td>
<td>3 632 256</td>
</tr>
<tr>
<td>Momentum equation</td>
<td>Segregated</td>
</tr>
<tr>
<td>Solver Type</td>
<td>Segregated</td>
</tr>
<tr>
<td>Convective scheme</td>
<td>QUICK</td>
</tr>
<tr>
<td>Turbulence equation</td>
<td>$k - \omega$ SST</td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>$k - \omega$ SST</td>
</tr>
<tr>
<td>Convective scheme</td>
<td>Upwind</td>
</tr>
<tr>
<td>Initial conditions</td>
<td>Family-based: BCInfow</td>
</tr>
<tr>
<td>Initial momentum</td>
<td></td>
</tr>
<tr>
<td>Initial turbulence intensity</td>
<td>1%</td>
</tr>
<tr>
<td>Initial eddy viscosity</td>
<td>1.0</td>
</tr>
<tr>
<td>Relaxation settings</td>
<td>Implicit-Explicit</td>
</tr>
<tr>
<td>Relaxation parameters</td>
<td>(0.9,0.9,1,0.25)*</td>
</tr>
</tbody>
</table>

Table 7.1 shows the details of the grid refinement study and figure 7.1 gives the extrapolations for the duct thrust coefficient and the radial force on the duct. It is observed that the grids do not provide a monotonic convergence neither for the duct thrust nor for the radial force. The computation of the convergence index
and the extrapolation of the exact solution are therefore affected by the oscillatory convergence. As a result, a value for the convergence index is found smaller than 0.5 (or not possible to determine) as shown with the label \( p = \{+1, 2\} \) which refer to the additional first-and-second order power fit \( \delta_{12} \) (section 2.6). In view of the poor data distribution, a large safety factor is used in the calculation of the uncertainty, which results to be 7.7% for the duct thrust and 9.5% for the radial force. Table 7.1 lists the discretization error, numerical uncertainties and CPU time for the six grids. The error in \( K_{TD} \) does not fall below 2% even for the finest grid, whereas for \( F_{RD} \) it is 3% at minimum. However, in view of the computational time, the finest grid is not an option so grid 3 is chosen for all operating conditions which gives an error of 3.28% in the thrust and 4.09% in the radial force. The uncertainties are high with a 9.7% and 10% respectively. The cause for the poor convergence in refinement is usually a questionable grid similarity\[31\]. Furthermore, as described in section 2.4, the refinement in the interpolation of the propeller force to the RANS grid is regulated by the number of steps in the Volume Intersection method. For this study the same number of refinement steps are used for all grids, but the distribution of body force is dependent on the RANS grid density. Hence it is recommended that a refinement in the interpolation steps is considered together with the mesh refinement; this could lead to improvements in the grid convergence study with a more satisfactory (monotonically) convergence. Finally, table 7.2 sums up the characteristics and settings for the final grid chosen for the computations.

7.3. BEM and RANS-BEM solutions

The results for the RANS-BEM computations for the propeller loading, duct loading and efficiency are compared with the experimental results and the BEM computations (here the Procal base calculation is considered) in the open water diagram and radial force in figures 7.2. For the radial force there are neither experimental nor full RANS solution so a proper validation is not possible. In addition, table 7.3 and 7.4 list the relative error with reference to the experimental results. The hybrid computations lead to a significant improvements in the prediction of the duct thrust. At the design condition \( J=0.5 \) there is a drop from an error slightly less than 30% in the BEM, to an error of 4.17% for the hybrid approach. The latter value, considering the 9% numerical uncertainty range, is in agreement with the experimental value. The duct thrust has a maximum of 8% error, except at \( J=0.7 \) when a meaningless large error is the result of the absolute value of the thrust being close to zero. However, the propeller thrust and the torque are overpredicted by the hybrid approach. Considering the design condition, there is a 9.97% overprediction of the propeller thrust and 6.07% overprediction of the torque. However, since there is an overprediction in both \( K_{TP} \) and \( K_{Q} \) (and the duct thrust is in good agreement), the open water efficiency \( \eta_0 \) is also better predicted in the hybrid approach. \( \eta_0 \) has an accuracy lower than 3% at \( J<0.6 \). The error becomes bigger for the light loading condition.

As concerns the radial force, it is noticed that it matches between the BEM and RANS-BEM computations for \( J=0.7 \). In the same condition, also the propeller thrust and torque match, which confirms that the radial force on the duct determines the induced velocities. The Procal reference solution predicts a larger radial force at high loading, which leads to a lower propeller loading. However, the lack of validation material does not allow to draw conclusions on the accuracy of the predicted value. It is expected that the RANS solution
is more accurate, but the occurrence of flow separation at the blunt 37 duct trailing edge poses questions on the prediction of propeller loading with an axisymmetric RANS-BEM approach, as discussed in section 7.4. The reason for the overprediction of the propeller loading in the RANS-BEM is investigated in the coming section. First of all, it is relevant to check if the potential flow solution is correctly coupled to the viscous solver, by comparison of the induction velocities. Then, the pressure distribution are analyzed to have a better understanding of the misprediction.

### 7.3.1. Open water induction

Before the analysis of the pressures and flow details, it is useful to check if the coupling procedure is correct. As suggested for instance by Starke[9], the coupling is checked by comparison of the open water induction velocity. The condition for a correct coupling is

\[ V_{ind}^{RANS} = V_{ind}^{BEM}, \]  

with \( V \) being the velocity in a location upstream of the propeller.

The induced velocities for the BEM are directly computed, while for the RANS+body-force the solution provides a total field (which is the field that is measured in the open water tests also). In order to get the induced velocities the effective wake has to be taken into account:

\[ V_{eff} = V_{tot} - V_{ind} \iff V_{ind} = V_{tot} - V_{eff}, \]  

When in open water condition the effective field is the uniform, undisturbed inflow so the calculation of the induced field for the RANS becomes straightforward. As concerns the boundary element method, there are source and dipole distributions at the panels on the blade surface. It is shown (Starke[9]) that for a correct check of the coupling procedure, only the contributions of the dipoles should be considered. This is because the source strength is determined from the impermeability condition and so the sources are responsible for modeling the blade thickness. The induced velocities both in axial and radial directions are checked at two location upstream of the propeller, at planes parallel to the propeller plane at \( x/R = 0.3 \) and \( x/R = 0.5 \). The right figure 7.3 gives a view of the propeller and a section of the duct, together with the location upstream of the propeller where the induced velocities are compared. The left figure provides the velocity profiles between the hub radius \((0.167R)\) and the propeller radius, which is the same at every angular position because there is axisymmetry in open water condition. This analysis is executed for an advance ratio of \( J = 0.5 \). As concerns the BEM solution, there are both cases when only the dipole strength is considered or also the source strength is included. Both radial and axial velocities are negative (except \( V_R \) very close to the hub), which means that the flow is accelerated in downstream and centripetal directions. The axial component is larger in absolute value for the downstream plane whereas the radial component viceversa is larger more upstream.

As a first conclusion, it noted that the larger differences between the BEM and the RANS occur close to the hub radius because the RANS solution has a no-slip condition while tangential velocity is allowed in the potential flow. Secondly, as expected, the BEM solution including source strengths underpredicts the velocity because of the blockage effect due to the physical presence of the blade. Limiting the comparison to the *dipoles only* case, there is only a negligible discrepancy with the *RANS+body_force* close to the hub for \( V_r \) at the plane at \( x/R = 0.3 \). This is not a symptom of an incorrect coupling, but rather just the consequence of the difference in boundary condition at the hub. It is then concluded that the induced velocities are consistent, and the RANS-BEM coupling is correct. Finally, it is worthy to remember that the velocities compared here are induced by the propeller+duct system. It would be useful for motivating the overprediction in propeller loading, to compare the induced velocities by the duct only. However, this investigation is not possible because there is no method to separate the induction due to the propeller and the duct in the RANS computations.

| \( J \) | \%\( \Delta K_{TP} \)(%) | \%\( \Delta K_{TD} \)(%) | \%\( \Delta 10K_Q \)(%) | \%\( \Delta \eta_0 \)(%) | \( J \) | \%\( \Delta K_{TP} \)(%) | \%\( \Delta K_{TD} \)(%) | \%\( \Delta 10K_Q \)(%) | \%\( \Delta \eta_0 \)(%) |
|---|---|---|---|---|---|---|---|---|---|---|
| 0.20 | 0.52 | 15.18 | 1.57 | 8.46 | 0.20 | 10.17 | 4.99 | 6.18 | 1.62 |
| 0.30 | 1.11 | 18.04 | 2.71 | 8.80 | 0.30 | 9.93 | 5.57 | 6.06 | 2.01 |
| 0.40 | 1.57 | 21.66 | 2.76 | 8.70 | 0.40 | 9.49 | 4.84 | 5.88 | 2.10 |
| 0.50 | 0.94 | 29.82 | 2.18 | 9.18 | 0.50 | 9.97 | 4.17 | 6.07 | 2.39 |
| 0.60 | 3.43 | 61.87 | 0.81 | 12.23 | 0.60 | 11.61 | 7.96 | 6.78 | 3.96 |
| 0.70 | 15.89 | 3631.67 | 8.19 | 27.87 | 0.70 | 18.68 | 385.29 | 10.05 | 9.96 |
| 0.80 | 59.32 | 113.04 | 27.79 | 219.90 | 0.80 | 41.09 | 67.0 | 18.34 | 77.92 |

Table 7.3: Relative change wrt experimental. BEM Ka4-70 in 37 duct.

| \( J \) | \%\( \Delta K_{TP} \)(%) | \%\( \Delta K_{TD} \)(%) | \%\( \Delta 10K_Q \)(%) | \%\( \Delta \eta_0 \)(%) | \( J \) | \%\( \Delta K_{TP} \)(%) | \%\( \Delta K_{TD} \)(%) | \%\( \Delta 10K_Q \)(%) | \%\( \Delta \eta_0 \)(%) |
|---|---|---|---|---|---|---|---|---|---|---|
| 0.20 | 0.52 | 15.18 | 1.57 | 8.46 | 0.20 | 10.17 | 4.99 | 6.18 | 1.62 |
| 0.30 | 1.11 | 18.04 | 2.71 | 8.80 | 0.30 | 9.93 | 5.57 | 6.06 | 2.01 |
| 0.40 | 1.57 | 21.66 | 2.76 | 8.70 | 0.40 | 9.49 | 4.84 | 5.88 | 2.10 |
| 0.50 | 0.94 | 29.82 | 2.18 | 9.18 | 0.50 | 9.97 | 4.17 | 6.07 | 2.39 |
| 0.60 | 3.43 | 61.87 | 0.81 | 12.23 | 0.60 | 11.61 | 7.96 | 6.78 | 3.96 |
| 0.70 | 15.89 | 3631.67 | 8.19 | 27.87 | 0.70 | 18.68 | 385.29 | 10.05 | 9.96 |
| 0.80 | 59.32 | 113.04 | 27.79 | 219.90 | 0.80 | 41.09 | 67.0 | 18.34 | 77.92 |

Table 7.4: Relative change wrt experimental. RANS-BEM Ka4-70 in 37 duct.
7.4. Pressure distributions on the propeller and on the duct

Given that the coupling is working properly, the pressure distributions at the duct and at the blade are analyzed. Figure 7.4 presents the mean pressure distribution at the duct surface for \( J=0.2 \) and \( J=0.5 \). Equally to chapter 5, there are four sets presented: the mean value for the reference BEM (i.e. Procal only), the coupled distribution for the BEM and RANS sides of the coupling, and the full RANS solution. In view of the coupling, the radial force, which is the largest component of the force acting on the nozzle, is the same for the coupled solutions. To begin with, the major differences occur at the duct inner surface, where the pressure is lower, while at the outer side the pressures are in good agreement, except for the BEM solutions at \( J=0.5 \) where there is a pressure recovery at \( x/c > 0.8 \). This is the consequence of the modified duct geometry for the potential flow calculations. Secondly, a comparison of the BEM reference and coupled solutions reveals that the pressure is larger at the inner side for the BEM coupled, which is coherent with the observed overprediction of propeller thrust and torque. Taking a look at the full RANS distribution used as validation case, it is rather oscillatory because, as explained in chapter 3, this is obtained by averaging over a limited number of angular slices. Comparing the full RANS and the RANS coupled distributions, the former shows a region of lower pressure in correspondence of the propeller location at the inner duct surface, from \( x/c = 0.4 \) to \( x/c = 0.6 \) approximately, which relates to the development of the tip leakage vortex. Contrarily, upstream of the propeller the pressure is larger for the full RANS than the RANS coupled both at \( J=0.5 \) and \( J=0.5 \). Remarkably, there is a plateau in the RANS coupled pressure just aft of the propeller location, which extends from \( x/c = 0.6 \) to \( x/c = 0.8 \) in design and high loading. Finally, towards the trailing edge, for \( x/c > 0.9 \) the pressure in the RANS solutions is almost constant, which is the effect of flow separation at the duct inner side seen for all advance ratios for the blunt 37 duct.

Regarding the overprediction of propeller loading in the hybrid approach, the lack of integrated values for the radial force from the full RANS limits the conclusions on the accuracy of the prediction of the duct loading. The plots of the pressure distributions show regions of both higher and lower pressure between full RANS and RANS coupled, hence there might be a balancing effect such that the radial force is matched reasonably good, but this cannot be confirmed with certainty because the full RANS distributions are rather uncertain in view of the few angular positions used to derive the mean values (chapter 3). For future work, in order to set a light on possible misprediction of duct loading, it is then important to have a value of the radial force acting on the duct for validation.

In addition, the chordwise pressure distribution on the blade is plotted and validated against the full RANS solution. Figures 7.5 give the results for the advance ratio of \( J=0.5 \) and radial position of \( r/R = 0.5, 0.9 \) (more distributions are given in the Appendix). The distribution used for validation are oscillatory, likely due to interpolation issues, which is more visible at the larger radii. The potential flow solutions show a major pres-
sure recovery at the blade trailing edge, which is present to a lesser extent in the full RANS. The differences between the reference and coupled BEM solution are seen most at the pressure side at \( r/R = 0.9 \): the coupled solution has a larger pressure which relates to the aforementioned overprediction of \( K_T \). Similarly to the 19A duct case, at the larger radii there are the more discrepancies with the full RANS distribution. Closer to the tip, the flow is influenced by the detachment of the tip leakage vortex, which is not modeled in a potential flow context.

The next section provides the analysis of the flow in the domain by contour plots and streamlines of the velocity. Also, the flow separation occurring at the duct inner surface is discussed, and considerations on the overprediction of propeller loading are presented.

### 7.5. Contour plots

Contour plots of the body force in axial direction and the pressure coefficient are given in figures 7.6 for an operating condition of \( J=0.5 \) on the X-Z plane. The streamlines of the velocity are also superimposed. For all operating conditions, there occur flow separation at the blunt trailing edge with a region of recirculating flow. The streamlines downstream of the duct reattaches at \( x/R \approx -0.7 \) hence the recirculation region extends for roughly one fifth of the duct length downstream of it. The contour plot of the body force in axial direction displays the minimum at the center of the body force distribution. As for the pressure coefficient, there is a drop in pressure at the inner duct surface when the flow is accelerated. Furthermore, there is a region of almost constant pressure in the separated region at the trailing edge (a yellow contour) with a value of pressure coefficient that approaches zero. Just before the separation, a (green) area of almost constant pressure is
present.

When a lighter loading condition of \( J = 0.8 \) is considered, flow separation occurs at the outer surface of the nozzle. The extent of flow separation is visible in figures 7.7 where the dimensionless vorticity magnitude \( \omega' \) and the dimensionless axial component of the axial velocity \( V_x/V_0 \) are plotted. Since the outer surface of duct 37 is less inclined than the case of the 19A duct, the recirculating flow region extend to a less extent: the length of the recirculating region is less than one half of the duct length while it is seen that for the 19A duct it covers the entire surface. The vorticity is large at the nozzle inner surface where the boundary layer develops and at the shear layers where flow separation occurs, both at the inner and outer duct sides. Regarding the axial velocities, it is always negative except in the recirculation regions.

While separation occurs at the nozzle outer surface at large \( J \) also for the 19A duct, the flow separation seen at the duct inner surface is not present for the 19A duct. When looking at the radial forces for the duct 19A, the larger differences with the full RANS solution happen exactly at large \( J \), when there is separation. Moreover, it was observed by Willemsen[8] that the flow pattern in the recirculating region differ in the full RANS solution with the angular position. It is then significant that the unsteadiness in the recirculating flow pattern is lost when the RANS-BEM solution is analyzed. This could be the cause for a misprediction in the duct radial force, hence induced velocities, which leads to the overprediction of propeller loading resulting from the hybrid approach. This hypothesis cannot be verified because of the limited validation material available, hence for future work it is then recommended to compare the radial force with full RANS solutions.
7.6. **Summary**

The chapter has presented the results for the hybrid RANS-BEM computations of the Ka4-70 propeller in 37 duct. The grid refinement study led to higher uncertainties (10% for the selected grid) because of the non-monotonic convergence of the dataset. This taken into account, the hybrid computations improves the prediction of duct thrust through the whole open water diagram in comparison to the Procal computations, which showed a constant overprediction. The open water efficiency has a 2% error with respect to the experimental data for the design condition and high loading. Furthermore, after an analysis of the open water induction velocities which confirmed the correct coupling of RANS and BEM solver, the pressure distributions are compared. They show that the potential flow solution is limited by the large modification in the duct shape, leading to unphysical pressure recovery. A comparison of the hybrid result with the full RANS distribution used for validation is not conclusive to motivate the overprediction of propeller loading. Finally, the analysis of the the contour plots shows that there is flow separation at the duct inner surface for all values of J. The differences in the separation points and extent of the recirculation region between the axisymmetric solution and a 3D full RANS approach are given as a possible cause for the misprediction of duct radial force (hence induced velocities) to be investigated when more complete validation material will be available.
Analysis of the tip leakage vortex and flow through the gap

In this chapter the CFD computations of the Ka4-70 propeller in 19A duct carried out at MARIN are analyzed with focus on the tip leakage and the flow in the gap between the propeller and the blade. Section 8.2 provides validation of the prediction for the detachment of the vortex by a comparison with the cavitation observations, as well as the development of the vortex more downstream is studied in section 8.3. Section 8.4 explains the interaction with the tip vortex and finally in section 8.5 the flow in the gap between the blade and the nozzle is considered.

8.1. Introduction
As illustrated in chapter 1, in the gap between the blade and the nozzle a vortex (namely the tip leakage vortex) develops and detaches at the suction side of the blade around mid-chord position with respect to the chord length at the blade tip. The tip leakage vortex is a strong vortex due to the combined effect of the finite pressure difference between the blade sides and the relative rotation of the nozzle and the propeller. It influences the loading on the duct-propeller system and the cavitation inception in view of the pressure drop at the vortex core. The scope of this chapter is to investigate the vortex structure and provide guidelines on possible modeling of the tip leakage vortex in a potential flow context. In order to do so, the full RANS CFD calculations (MARIN,2012[11]) on the Ka4-70 propeller in 19A duct are analyzed as well as the cavitation observations in the model test. The limited experimental and CFD data available for the case of duct 37 do not allow for an equal study for this geometry.

8.2. Tip leakage vortex detachment
The CFD solution is compared with the experimental observation to check if the numerical solution predicts correctly the location where the vortex is formed. The detachment of the tip leakage vortex is affected by blade loading, relative rotation and the geometry of the blade at the tip. It is proven (for instance in Dreyer et al. [19]) that the gap-ratio (which is the ratio of the gap width to the local blade thickness) highly influences the formation of the vortex. For the CFD calculations a gap of 1.00 mm is used while in the experiments a gap width of 1.01mm is set. Furthermore, for the CFD calculation the blade trailing edge was rounded ([11]) to ease the meshing procedure. The difference in gap width is negligible, while the difference in blade trailing edge is assumed not to influence the detachment of the vortex which occurs much before the blade trailing edge. Therefore, small differences in the gap ratios between experiments and CFD are negligible.

Figures 8.2 show a comparison between the experimental observations and the CFD solution for what concerns the detachment angle of the vortex at J=0.5. The view is from the side of the nozzle, the duct is transparent in the experiments and the blade tip profile is shown in the CFD. For the CFD data the vortex is visualized by plotting the isosurface of the (dimensionfull) Q-factor at \( Q = 10^5 \text{ m}^{-2} \). This value is selected such that only the leakage vortex is visualized. For the experimental observation the cavitation pattern is considered to draw considerations on the vortex detachment. For this study the observations for a cavitation number of \( \sigma = 1.3 \) are used. At the blade tip there is a vast region of cavitation which develops in the form of a sheet cavitation from the pressure to the suction side of the blade and a cavitating vortex more downstream. A closer look
at the cavitation pattern reveals that the vortex is located at the edge of the cavitation region, so this is the position taken for the measurement of the detachment angles.

Both for the experimental observations and the CFD plots, there is an uncertainty in the estimation of the vortex position, which takes into account the uncertainty for the visual inspection. In addition, for the experimental observations the blade-to-blade variations in cavitation pattern as described in chapter 3 are included. The CFD predicts a slightly smaller angle with respect to the chordline, but the value is within the uncertainty range. For a loading condition of $J=0.2$ (plots are given in Appendix), the same conclusion is drawn, with the detachment angle that increases for the CFD to 9.5°. This behavior is expected because for the higher loading the pressure gradients that drives the flow through the gap is larger. In addition, for the highest loading condition there is another vortex detaching around midchord position, a secondary structure that merges with the primary vortex.

Moreover, the detachment location is investigated, i.e. at which chordwise position the tip leakage vortex detaches. The side view is not suitable for this purpose because in the cavitation observations the position of the blade Leading Edge (LE) is shadowed. Therefore, the front view of the system is considered. Figures 8.2 show the CFD and cavitation results with the same view angle. The arclength of the blade at tip (as seen from this angle) is measured and set to unity so the location where the vortex detaches is given as fraction of it. The CFD predicts a vortex detachment closer to the leading edge. This is observed also for the high loading condition of $J=0.2$, as in table 8.2, when however the vortex detaches closer to the leading edge.

Finally, a remark is recommended for the comparison of CFD and experiments. The numerical solutions for the wetted flow are compared here with experimental cavitation observations. A more consistent comparison should consider numerical solution of cavitating flow (RANS or LES), especially because it was proved (for instance by Xia[22]) that tip cavitation largely affects the flow behavior at the gap flow for CFD calculations. However, it is seen that experimental observations for higher values of cavitation number result in a decrease of the extent of the cavitating region at the tip, but the location and angle of detachment are not strongly affected. It is then accepted that a comparison limited to the detachment location and angle is also valid for the available wetted-flow CFD solutions.

8.3. Tip leakage vortex trajectory

By an analysis of the full RANS computations, the development of the vortex is investigated. In this section the focus is on the trajectory of the tip leakage vortex after its detachment. The trajectory in the $x-\theta$ plane is studied in this section, where figure 8.3 shows the extended area of a cylindrical section of the propeller-duct system at a radial position of $r/R = 1$, with $R$ being the propeller radius. The contour plot of the dimensionless vorticity magnitude $w^*$ is plotted for angular positions from $−90°$ and $+90°$ and axial position from $x/R = −0.5$ to $x/R = +0.5$. So, the area shown covers the whole duct length in axial direction and half of it in circumferential direction. The propeller tip section is located between $x/R = −0.15$ and $x/R = +0.15$. In this figures, the horizontal and vertical axis are not equally scaled (for a better visualization) so the blade tip pitch angle is not actually as large as it appears from this plot. Upstream of the propeller the vorticity is zero, while downstream of the propeller the effect of the presence of the tip vortex in the wake results in the green streaks of larger vorticity at $x/R < −0.2$. Furthermore, there is a region in red of large vorticity in the blade passage. This is a region of large velocity gradients due to the leakage flow and the roll-up of the tip leakage vortex. In addition, there is a kink in the outer edge of the red of higher vorticity region. This is an indication that the slope of the trajectory of the tip leakage vortex changes abruptly. This change occurs at the mid-chord position. A change in the trajectory of the tip leakage vortex is observed in literature for the simplified case of a hydrofoil in proximity of a confinement wall. Dreyer[19] carried out a study of the effect of gap width in the development of the leakage vortex. The kink in the trajectory moves towards the blade tip trailing edge for larger gaps, whereas for smaller gaps the kink is observed at midchord position as it is in the CFD solutions studied here. Dreyer argued that the change in trajectory is due to the influence of the blade wall. When the vortex is close to the blade, both the duct confinement and the blade wall affect the vortex dynamics, but when the vortex gets further from the blade, only the nozzle surface influences the vortex motion. Although the interpretation in [19] is consistent with a vortex image analysis, it does not explain why the change in trajectory occurs abruptly, hence the kink in the vorticity contour. Nevertheless, it is seen that the change in trajectory occurs at the same mid-chord location where secondary structures seem to detach from the blade tip (section 8.5 and particularly figure 8.8). Therefore, it is likely that the appearance of secondary structures is the cause for the change in tip leakage vortex trajectory in the $x-\theta$ plane.
Figure 8.1: J=0.5 Ka4-70 in 19A duct. Tip leakage vortex detachment angle from full RANS computations and experimental observations. Experimental data refer to cavitation number \( \sigma = 1.3 \).

Figure 8.2: J=0.5 Ka4-70 in 19A duct. Tip leakage vortex detachment location from full RANS computations and experimental observations. Experimental data refer to cavitation number \( \sigma = 1.3 \).

<table>
<thead>
<tr>
<th>Tip leakage vortex detachment angle (degrees)</th>
<th>J=0.2</th>
<th>J=0.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>9.5 ± 0.5</td>
<td>8.2 ± 0.5</td>
</tr>
<tr>
<td>Experimental</td>
<td>9.8 ± 0.8</td>
<td>9.2 ± 0.8</td>
</tr>
</tbody>
</table>

Table 8.1: Predicted and experimental angles at which the tip leakage vortex detaches with respect to the projected chord length at the blade tip.

<table>
<thead>
<tr>
<th>Tip leakage vortex detachment location (fraction of arclength at the tip)</th>
<th>J=0.2</th>
<th>J=0.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>0.065 ± 0.03</td>
<td>0.155 ± 0.03</td>
</tr>
<tr>
<td>Experimental</td>
<td>0.14 ± 0.03</td>
<td>0.33 ± 0.03</td>
</tr>
</tbody>
</table>

Table 8.2: Location at which the tip leakage vortex detaches according to the front view of the CFD results and experimental observations.
8.4. Interaction of tip leakage vortex and tip vortex
The tip leakage vortex detaches at small chordwise position. When the tip of the blade has a large chordlength, the vortex will interact with the trailing tip vortex. Differently, when the propeller has a small chord length at the tip, one vortical structure will develop. The objective is therefore to check what is the vortex-vortex behavior for the Ka4-70 propeller geometry in the 19A duct. The contour plots of vorticity (and Q factor, section 2.7) are produced for four planes at increasing angular position as shown in figure 8.4. The first plane intersects the blade tip trailing edge while planes 2 to 4 investigate the vortex interaction more downstream.

The contour plots of the dimensionless angular component of the vorticity \( w'_\theta \) are given in figure 8.6 for planes 1 to 4. The region shown is a zoom-in region of the \( x/R \) planes at large radii and in correspondence of the location of the vortices (hence at increasing \( x/R \) with the plane number). In plane 1, the section of
the blade is visible in white to the right of the plot, and a region of large vorticity is visible in red. In plane 2 the elongated region of higher vorticity display two locations where the vorticity is maximum: region \(a\) is attributed to the tip leakage vortex whereas region \(b\) relates to the tip vortex detaching from the blade Trailing Edge (TE). It is significant that the vorticity has the same sign for both vortices, meaning that they are co-rotating. At the more downstream planes 3 and 4 the two vortices interact: the tip leakage vortex \(a\) is displaced at slightly lower radial position while the tip vortex is clearly moved to larger radial position and eventually merges with the duct boundary layer. The behavior of the vortices can be explained by a vortex image system, in a similar fashion as done by Chen\[37\]. The top figure\[8.5\] shows the system composed by the two co-rotating real vortices (\(a,b\)) and the two image vortices required for the vortex image analysis (c,d). The four vortices induce velocities on each another, with for instance velocity \(V_{ba}\) being the velocity induced on vortex \(a\) by vortex \(b\). The analysis carried out here is only qualitative, so the direction is significant rather than the magnitude of the velocities. The bottom figure in \[8.5\] gives the resulting velocity vector which explains the upward motion of the tip leakage vortex and its merging with the duct boundary layer. For the sake of completeness, a zone of negative angular vorticity is seen in blue very close to the duct wall. This is due to the flow reversal with positive axial velocity induced by the vortex.

The analysis of the vortex-vortex interaction shows that the tip leakage is retained downstream, hence the modeling of the trailing wake behind the propeller in the boundary element method should represent the pitch and trajectory (in terms of contraction of the wake) of the tip leakage vortex, and not of the tip vortex as it is the case of open propellers. Finally, the study of the vortex interaction is here carried out by looking at the \(x-R\) planes because, as visible in figure \[8.1\], the vortex line develops in circumferential direction mainly. Nonetheless, the vorticity has a nonzero axial component at the vortex (while the radial component is negligible), which is not investigated here. To have an even better view of the vortex behavior it would be beneficial to analyze the vortex in a plane orthogonal to the vortex line. However, this task would involve two difficulties: first of all the vortex trajectory must be known a-priori; secondly, the contribution of additional vorticity component due to the relation between inertial and rotating reference system must be considered.
Due to the extra post-processing required, this is not considered in this work.

### 8.5. Flow in the gap between the blade and the nozzle

A further analysis to give guidelines for the boundary element method and provide more understanding of the leakage flow is carried out with focus on the flow through the gap. For this purpose, angular planes are again considered, but in this case five planes are extracted from the CFD solution, from an angle of $-21.2^\circ$ to $+21.2^\circ$ corresponding respectively to the intersection with the leading edge and the trailing edge at the blade tip as shown in figure 8.7. By zooming-in at the region of the blade tip, the flow at the gap is investigated.

Figure 8.8 shows the contour of the dimensionless axial velocity $V_x/V_0$, with $V_0$ the freestream inflow velocity, for the plane at zero degree angle. The streamlines of the axial-radial velocity are also shown and the plots for the other angular planes are given in the Appendix. The velocities plotted here are defined in the absolute (i.e. non-rotating) reference system.

First of all, the streamlines show the tip leakage vortex which is located approximately at the same radial position as the blade tip, slightly more upstream at the suction side of the blade. Furthermore, the visualization is such that the streamlines seem to impinge into the propeller blade at the suction side and leave from the blade at the pressure. However, the velocity as a major angular component which is oriented out of plane that is non-zero at the blade surface in view of the rotation of the propeller. A closer look at the gap reveals that there is nearly zero axial velocity in the gap. Remarkably, the gap is completely obstructed by the tip leakage vortex. This condition is observed for the angles of $0^\circ$ and $+10.6^\circ$. At the leading edge the flow has negative axial direction over the whole gap while at $-10.6^\circ$ there is a backward flux only closer to the duct surface. Finally, at the plane at $21.2^\circ$ (intersecting the trailing edge) the tip leakage vortex gets further from the blade and complete obstruction does not occur. The same considerations are drawn for the higher loading condition $J=0.2$ but in this case there is a second vortex detaching around 60% of the chordlength which merges with the primary vortex before the blade trailing edge and contributes to the obstruction of the gap.

As a conclusion, the velocity in axial direction through the gap is non-zero towards the blade trailing edge, roughly zero at the midchord where the gap is obstructed, and again non-zero at the trailing edge when the tip leakage vortex is far from the blade. Therefore, the gap flow model for the BEM package should reproduce this behavior for the axial velocities.

---

1 Nonetheless, it is noted that if velocities were plotted in the non-inertial (rotating) reference system, there would be an additional component to the angular velocity $V_\theta$, which is oriented out of plane. Hence, the plots of the velocities like figure 8.8 are invariant for the reference system. The change of reference system affects the angular component of the velocity and the $X$ component of the vorticity. Therefore, as far as these components are not used, the contour plots will be the same for the rotating and non-rotating reference frame.
Figure 8.9: View of the mesh for the full RANS computations of the Ka4-70 propeller in 19B duct\cite{11} and zoom-in on the detail of the grid distribution at the blade tip. The grid is dense close to the walls (duct wall and blade surface) but it is coarser in the region of the domain where the vortex is observed to develop.

In addition, as shown in the zoomed-in rectangle in figure 8.8, at the blade tip the flow is mildly recirculating. This suggests that the shear layer at the tip is subjected to roll-up also at more downstream location than the chordwise position where the tip leakage vortex detaches. This is an indication of generation of secondary structures. This phenomenon is observed to occur at the mid-chord position, and likely is related to a change in the trajectory of the primary tip leakage vortex (section 8.3).

Finally, a consideration on the grid density is necessary. Figures 8.9 show an overview of the grid on the domain and a zoom-in on the tip of the blade. The grid is dense in vicinity of the propeller wall and the duct surface, but it coarsen far from walls, most in the region where the vortex is formed. These computations\cite{11} were indeed oriented to a good prediction of integral values (the propeller and duct loading) rather than investigating such flow details. Therefore, strong conclusions on flow details at small scales such as the roll-up of secondary structures are not safe. For future work, a more thorough analysis would require a more refined solution, for instance by mean of adaptive grid algorithms.

8.6. Summary

The chapter presents a detailed analysis of the full RANS calculation\cite{11}. The study of the detachment of the tip leakage vortex reveals a good agreement between the experimental data and the CFD for the detachment angle, but an underprediction for the detachment location. The tip leakage vortex develops downstream with a change in the slope of the trajectory in the $x-\theta$ plane, likely due to the appearance of secondary structures. When getting closer to the blade tip, the tip leakage vortex interacts with the tip vortex: the latter is displaced at larger radii and merges with the duct boundary later. Hence, the pitch of the tip leakage vortex is the reference pitch for the tip of the wake surface attached to the propeller in the boundary element method. Finally, the analysis of the gap flow shows that the flow is obstructed by the tip leakage vortex at the mid-chord position, but not at the leading edge (where the vortex has not developed yet) nor at the trailing edge (where the vortex gets far from the blade).
Conclusions and recommendations

In this study a boundary element method (BEM) and a hybrid RANS-BEM method for the prediction of the performance of ducted propellers are analyzed with respect to the open water performance of two propulsors, which have the same propeller geometry (Ka4-70) but differ in the annular nozzle surrounding the propeller (duct 19A and duct 37). The accuracy of the methods for the open water loading is determined by a comparison with the experimental results while a more detailed comparison of the pressures on the blade and duct surfaces with full RANS data allows to understand the limitations of the models. In addition, the analysis of the available full RANS computations gives more insight on the leakage flow and the flow pattern at the gap between the blade tip and the nozzle wall. This turns useful to give guidelines on the modeling of such phenomena in a potential flow context. The conclusions of this study are given separately for the two computational methods and the leakage flow analysis.

9.1. Performance prediction using the BEM method

The first objective of this research was to investigate the accuracy of the boundary element method Procal for the two test cases. In view of the required change in the geometry to have a sharp trailing edge, a new iterative scheme based on a pressure-equality condition is applied. Furthermore, a sensitivity study on the transpiration velocity model implemented in Procal is carried out for the first test case. Chapters 4 and 6 lead in the following conclusions:

• The new iterative scheme for the radial position of the trailing edge is a valid alternative to determine a suitable duct geometry without the need of a previous sensitivity study or a-priori knowledge of the flow behavior (particularly the streamlines pattern at the duct trailing edge). The scheme is based on a pressure-equality condition. The recommended location for the condition of equal pressure at the inner and outer duct surfaces corresponds to the position of the original trailing edge. However, when larger changes in geometry are involved as in the case of the duct 37, a modified location must be considered because of the non-physical pressure recovery resulting from a potential flow analysis. As a conclusion, the scheme is expected to be applicable for other geometries, given that the user has some experience to predict a pressure recovery with a potential flow method.

• The boundary element method Procal shows satisfactory results for the in-design and the high loading conditions, but the prediction is poor for the larger advance ratios. In the former case, the propeller thrust and torque show a 6% error for the 19A duct and a 3% error for the duct 37. The duct thrust is seen to be overpredicted, particularly for duct 37, such that the open water efficiency has a 7% and 9% relative difference with respect to the experimental results for the duct 19A and 37 respectively. With advance ratios at light loading, flow separation at the duct outer side occurs, which is large for the 19A duct and milder for the 37. Separation is not modeled in the BEM and it is a limit for the applicability of BEM for $J>0.6$.

• The comparison of the pressure distributions at design condition between the BEM and the full RANS approach shows that there is a balancing effect for the duct thrust. At large $J$ the lack of flow separation results in an overestimated pressure at the outer duct surface hence the resulting large value of duct thrust.
• For the test case of the Ka4-70 propeller in 19A duct, there is an inconsistency in the hub diameter for the experimental data and the full RANS calculations used for validation, hence the effect of two hub diameters is investigated. The BEM solution shows a difference of 3% at maximum between the two.

• From the results of the sensitivity study for the transpiration velocity model in Procal, it is concluded that changes in the user-prescribed boundary layer displacement thickness at the trailing edge affects mostly the radial force acting on the duct. The radial force is influenced 1.5 times the propeller thrust and 4 to 5 times the duct thrust. Since the transpiration velocity model is enforced for the coupling procedure in the hybrid approach, it is concluded that the force acting on the duct in radial direction must be considered for the coupling procedure.

9.2. The hybrid RANS-BEM method

The use of a hybrid RANS-BEM approach allows to solve the viscous flow effects at the duct. The propeller is not physically present in the RANS computations, but it is substituted with a body force distribution whose strength comes from the BEM method. The coupling is based on an iterative procedure to match the radial force acting on the duct between the BEM part and the RANS+body-force part of the coupling. Therefore, the propeller force distribution interpolated to the RANS at the last iteration is such that the duct radial force is the same. The conclusions on this method follow from chapters 5 and 7 where the hybrid approach is used for the two test cases.

• The RANS-BEM leads to a good prediction of the duct thrust, with an error below 4% at design condition for both duct 19A and duct 37. The improvements in comparison with the BEM method are significant especially at the larger advance ratios, where the viscous simulations allows for taking the flow separation into account. The error in duct thrust is limited to only 1% and 7% for the 19A and 37 duct respectively at \( J = 0.8 \). Furthermore, for the first test case also the propeller thrust and torque are better predicted with the hybrid scheme, with a drop from the aforementioned 6% error in the BEM, to a value of 4% in the thrust and 3% in the torque. As a result, the open water efficiency for the high loading and design condition is predicted with an accuracy around 2%. This is a satisfactory result for the hybrid RANS-BEM approach as it is the same error of the more expensive full RANS computations (chapter 3).

• The analysis of the pressure distributions shows that the axisymmetric RANS-BEM solution differs from the averaged full RANS solution in an underprediction upstream of the propeller and the absence of the pressure peak due to the tip leakage vortex in the blade passage. These two effects balance each other such that the radial force, which is the largest component of the duct loading, is well predicted at \( J \leq 0.6 \). For larger advance ratios, the pressure distributions from the full RANS and RANS-BEM differ largely at the duct outer side: this is the effect of a different flow separation pattern between the 3D case and the axisymmetric RANS+body force.

• When the more challenging duct 37 geometry is considered, the open water efficiency is predicted with a 3% to 4% accuracy in the high and mid-loading conditions. However, both propeller thrust and torque are overpredicted. The thrust is overpredicted by a 10% and the torque by a 6-7%. For this geometry, in addition to the flow separation at the duct outer surface at large \( J \), there occur also separation at the inner duct surface close to the blunt trailing edge for all the loading conditions. The extend of flow separation in the hybrid computations is a possible reason for the overprediction of duct loading.

9.3. On the analysis of the tip flow

The analysis of the tip gap flow was presented in chapter 8 with the scope of comparing the numerical full RANS solution with the experimental cavitation observations and provide guidelines for the potential flow modeling. The following conclusions are obtained from this task:

• The Ka4-70 propeller has a large chordlength at the tip. Because of the combined effect of the pressure difference between the two sides of the blade and the the relative rotation of the nozzle and the propeller, a tip leakage vortex detaches at small chordwise position. The tip leakage vortex detaches closer to the leading edge for higher loading, and for \( J = 0.2 \) even a second vortex is seen around mid-chord, which merges with the first vortex more downstream. Through a comparison of the cavitation observations with the CFD, it is concluded that the detachment location is overestimated in the full RANS while the detachment angle is within the uncertainty range. Furthermore, the tip leakage vortex
trajectory shows a change in the pitch of the vortex. This is likely related to the occurrence of secondary structures detaching from the blade tip.

- The leakage vortex interacts with the tip vortex in the propeller wake. The vortices are co-rotating and the tip vortex merges with the boundary layer at the duct wall as a result of the vortex-vortex interaction. Therefore, when modeling the wake sheet attached to the propeller in the BEM method, the pitch of the tip leakage vortex is representative for the pitch of the wake sheet at the tip.

- A detailed analysis of the gap flow by means of contour plots and streamlines of the velocity shows that the gap is obstructed by the vortex at the midchord position. Differently, at the tip trailing edge the vortex is relatively far from the blade so a flow in axial direction through the gap is seen. A gap flow model for the boundary element method should then be able to include the blocking effect of the vortex.

### 9.4. Recommendations for future work

Based on the results of this research project, there are some aspects that are interesting for future investigations. Therefore, the following indications for future work are recommended:

- It is suggested to carry out a comparison of the radial force resulting from the hybrid RANS-BEM approach for duct 37 with the validation values from full RANS, once available. This will allow to draw stronger conclusions on the overprediction of the propeller loading. For the same reason, an analysis of the flow separation at the inner surface should be object of future work.

- A rather large value of uncertainty of 7% for the duct force is obtained here for the hybrid method. This is the result of oscillations in the grid refinement study. For the RANS-BEM coupling, the propeller force is interpolated to the RANS grid with a recursive interpolation scheme, for which a constant refinement in the interpolation algorithm is considered in this work. For future investigations, it is suggested to carry out refinement study of the recursive steps as presented in chapter 5 for different grid densities. Then, evaluate if a refinement in the interpolation levels is necessary together with a mesh refinement to lower the uncertainty.

- The iterative scheme for the trailing edge position based on a pressure-equality condition was applied for the duct 19A and 37 with satisfactory results. The scheme was developed in this work with the goal of choosing a proper trailing edge position for the paneling of the duct in Procal. Since a change in the effective duct camberline is obtained with the transpiration velocity model, it is also possible to obtain the same equal-pressure condition by matching the source strength in the transpiration model. This task was not carried out and it is then suggested for future work.

- As concerns the analysis of the tip leakage flow, in chapter 8 the wetted flow computations are compared with the experimental cavitation observations. The effect of cavitating flow at the blade tip is therefore left out of the CFD calculations. In terms of integral quantities as the propulsor loadings and efficiency, the wetted flow analysis is satisfactory but for a detailed study of the gap flow, the occurrence of sheet cavitation at large radii or tip leakage vortex cavitation will affect the leakage flow. Therefore, the next step would be to analyze numerical simulations of cavitating flow. Finally, the full RANS calculations were given for one mesh and one setting for the turbulent model. The chosen grid is suitable for the prediction of the integral quantities (thrust and torque) but a grid refinement study concerning the strength, the size and the development of the tip leakage vortex was not carried out. Therefore, it is recommended to investigate the sensitivity to the mesh density. Particularly, it would be interesting to provide a detailed solution (for example using an adaptive grid) of the flow in the region between the propeller blade suction side and the detached tip leakage vortex, in order to understand if secondary structures are developing and how they interact with the primary vortex. Finally, the influence of the turbulent model should be investigated. That is expected to play a role in the gap flow pattern, where large velocity gradients are observed with the roll-up of the shear layer developed in the leakage flow.
The general solution for the potential problem for boundary element methods is derived by use of Laplace equation and Gauss divergence theorem. The Gauss divergence theorem for a vector field of the form $\Phi_1 \nabla \Phi_2 - \Phi_2 \nabla \Phi_1$ is given by:

$$\int_S (\Phi_1 \nabla \Phi_2 - \Phi_2 \nabla \Phi_1) \cdot n \, dS = \int_V (\Phi_1 \nabla^2 \Phi_2 - \Phi_2 \nabla^2 \Phi_1) \, dV, \quad (A.1)$$

that is known as Green’s first identity. Here $S = S_B + S_W + S_\infty$, the sum of body, wake and far-distance surfaces. Furthermore, let:

$$\Phi_1 = \frac{1}{r}, \quad \Phi_2 = \Phi,$$  \quad (A.2)

where $\Phi$ is the potential of the flow and $r$ the distance from the location at the surface and a point $P(x, y, z)$ in the domain $V$, as in figure A.1.

When $P$ lies in the domain $V$ (which excludes the internal volume of the body) $\Phi_1$ is not defined at $P$. Furthermore, except at $P$, the Laplace equation holds for both $\Phi_1$ ($\nabla^2 (1/r) = 0$ (easily proved in spherical coordinate) and for $\Phi_2$ by definition of potential for incompressible flow. To prevent the singularity at $P$, the point is excluded from the domain by setting a surrounding sphere of radius $\epsilon$ with normal unit vector pointing inward, as in figure A.1. So, substituting (A.2) in (A.1) and integrating also over the surface of the sphere, the right hand side in (A) vanishes:

$$\int_{S + \text{sphere}} \left( \frac{1}{r} \nabla \Phi - \Phi \frac{1}{r} \nabla \right) \cdot n \, dS = 0. \quad (A.3)$$

![Figure A.1: Illustration of potential flow problem. Katz,Plotkin[4](92)](image-url)
Introducing a spherical coordinate system, and assuming that the potential does not vary much in the small sphere, it is:

\[-\int_{\text{sphere}} \left\{ \frac{1}{r} \frac{\partial \Phi}{\partial r} + \frac{\Phi}{r^2} \right\} dS = -\int_{\text{sphere}} \frac{\Phi}{r^2} dS \approx 0 \]

(A.4)

which then provides:

\[\Phi(P) = \frac{1}{4\pi} \int_{S} \left( \frac{1}{r} \nabla \Phi - \Phi \frac{1}{r} \right) \cdot n dS.\]

When taking a point \(P\) on the body surface \(S_B\), the integration must be carried out over an emisphere of radius \(\epsilon\) whose volume is all in \(V\), out of the body surface. In this case, the integral (A.4) is halved, and so:

\[\Phi(P)_{|P \in S_B} = \frac{1}{2\pi} \int_{S} \left( \frac{1}{r} \nabla \Phi - \Phi \frac{1}{r} \right) \cdot n dS.\]

(A.6)

Now consider a point inside the surface \(S_B\). Equation A.3 holds, but without the integration over the sphere, because the point inside the body is actually out of the flow region of interest, and there is no need to overcome singular behavior. By multiplying for a factor \(\frac{1}{\epsilon^3}\) it is:

\[0 = \frac{1}{4\pi} \int_{S_B} \left( \frac{1}{r} \nabla \Phi - \Phi \frac{1}{r} \right) \cdot n dS\]

(A.7)

with \(\Phi_i\) the internal potential and \(n\) pointing outward of \(S_B\). Summing (A.5) and (A.7) leads to

\[\Phi(P) = \frac{1}{4\pi} \int_{S_B} \left[ \frac{1}{r} \nabla (\Phi - \Phi_i) - (\Phi - \Phi_i) \frac{1}{r} \right] \cdot n dS + \frac{1}{4\pi} \int_{S_B + S_{\infty}} \left( \frac{1}{r} \nabla \Phi - \Phi \frac{1}{r} \right) \cdot n dS.\]

(A.8)

Furthermore, for the thin wake surface, the gradient in normal direction is continuous, so the expression for the potential is modified to

\[\Phi(P) = \frac{1}{4\pi} \int_{S_B} \left[ \frac{1}{r} \nabla (\Phi - \Phi_i) - (\Phi - \Phi_i) \frac{1}{r} \right] \cdot n dS + \frac{1}{4\pi} \int_{S_B + S_{\infty}} \Phi n \cdot \nabla \frac{1}{r} dS + \Phi_{\infty}(P)\]

(A.9)

in which the undisturbed potential \(\Phi_{\infty}\) is constant for an inertial reference system, Katz & Plotkin[4] and is defined as:

\[\Phi_{\infty}(P) = \frac{1}{4\pi} \int_{S_{\infty}} \left( \frac{1}{r} \nabla \Phi - \Phi \frac{1}{r} \right) \cdot n dS.\]

(A.10)

To solve the potential flow problem, the singularity distribution is introduced. Defining the source \(\sigma\) and doublet \(\mu\) as:

\[-\mu = \Phi - \Phi_i\]
\[-\sigma = \frac{\partial \Phi}{\partial n} - \frac{\partial \Phi_i}{\partial n}\]

(A.11)

(A.12)

so equation (A.9) is re-written:

\[\Phi(P) = -\frac{1}{4\pi} \int_{S_B} \left[ \sigma \left( \frac{1}{r} \right) - \mu \frac{\partial}{\partial n} \left( \frac{1}{r} \right) \right] dS + \frac{1}{4\pi} \int_{S_B + S_{\infty}} \left[ \mu \frac{\partial}{\partial n} \left( \frac{1}{r} \right) \right] dS + \Phi_{\infty}(P),\]

(A.13)

where the inner product of gradient and normal vector is the partial derivative in normal direction. When the point \(P\) is on the surface \(S_B\) the same procedure, but considering integration over an hemisphere as in (A.6), leads to

\[\Phi(P)_{|P \in S_B} = -\frac{1}{2\pi} \int_{S_B} \left[ \sigma \left( \frac{1}{r} \right) - \mu \frac{\partial}{\partial n} \left( \frac{1}{r} \right) \right] dS + \frac{1}{2\pi} \int_{S_B + S_{\infty}} \left[ \mu \frac{\partial}{\partial n} \left( \frac{1}{r} \right) \right] dS + \Phi_{\infty}(P).\]

(A.14)

Equation (A.13) and (A.14) present the statement of the potential problem: finding the appropriate singularity distribution for which the potential fulfills the boundary conditions.
The Actuator Disk Theory for ducted propellers

In this Appendix the actuator disk theory for the ducted propellers is explained, as it is used to derive the relations for the induced velocities and the ideal efficiency given in chapter 1. The theory explains the working principle of ducted propellers and gives the most simple model of the system. The derivation presented here follows Kuiper[3], but it is integrated with the formulation for ducted propellers given by Oosterveld and Gent[2][38].

The Actuator Disk Theory is based on the laws of conservations of mass, conservation of impulse and the law of Bernoulli. For a generic control volume in the fluid, in view of the Newton’s law $F = \frac{d}{dt}(m_v)$, the forces due to impulse acting on the volume are given by:

$$F_{\text{impulse}} = \int \rho < \vec{v} \cdot \vec{n}> \vec{n} dA. \quad (B.1)$$

Equally, the forces on the control volume due to the pressure are provided by integration over the volume boundaries:

$$F_{\text{pressure}} = \int -p \vec{n} dA. \quad (B.2)$$

By summing up the two components, the external force acting on the control which is considered for the change in momentum is obtained:

$$F = \int -p \vec{n} dA + \int \rho < \vec{v} \cdot \vec{n}> \vec{n} dA. \quad (B.3)$$

The other fundamental equation which is used is the Bernoulli equation. For an incompressible, inviscid, steady flow, the Bernoulli function is constant over streamlines:

$$p_{\text{stat}} + \frac{1}{2} \rho V^2 = \text{const.} \quad (B.4)$$

With the conservation of impulse and the Bernoulli equation, the actuator model lets a simple analysis of the duct-propeller system. The model is presented in figure B.1. The uniform inflow has velocity $V$ and pressure $p_0$, the duct is retained while the propeller is substituted with a disk which provides a uniform loading by a pressure drop $\Delta p$ (the actuator disk). For the sake of this proof, let the duct be an accelerating duct type. Due to the combined effect of the propeller and the duct, the flow is accelerated at the location of the propeller by an induced velocity $v_1$ and in the wake by a velocity $v_2$. The first assumption in this method is the flow being inviscid, in order to apply Bernoulli. Secondly, the forces in radial and tangential direction exerted by the propeller and duct are neglected, as well as the velocities in radial and tangential directions. Third, the propeller is not physically present and the rotational effects are not accounted for. Therefore, the viscous and rotational losses are disregarded and only the axial losses in the wake are considered.

To derive the propeller thrust, the pressure jump at the disk is found by mean of the Bernoulli equation. When applied at two points located at the upstream section $A_0$ and just before the propeller plane, Bernoulli gives:

$$p_0 + \frac{1}{2} \rho V^2 = p + \frac{1}{2} \rho (V + v_1)^2, \quad (B.5)$$

94
Figure B.1: Actuator disk theory for the ducted propeller. The contour volume where the conservation of impulse and Bernoulli equation are applied is shown in red. From Kuiper[3].

while downstream of the propeller and in the wake it holds:

\[ p_0 + \frac{1}{2} \rho (V + v_2)^2 = p + \Delta p + \frac{1}{2} \rho (V + v_1)^2. \] (B.6)

Subtracting the two equations, the uniform pressure jump at the actuator disk is obtained:

\[ \Delta p = \frac{1}{2} \rho (2Vv_2 + v_2^2), \] (B.7)

which multiplied by the area of the propeller provides the propeller thrust:

\[ T_p = \frac{1}{2} \rho v_2 A(2V + v_2). \] (B.8)

As concerns the duct thrust, the conservation laws for mass and impulse conservation are used for the derivation. When the streamtube is considered as control volume, the pressure integral represent an external force:

\[ F = p_0 A_0 - p_0 A_2 - \int_{r_2}^{r_0} p_e 2\pi r dr + \rho V^2 A_0. \] (B.9)

The integral term is the integration of the pressures over the outer part of the volume. This can be split in two components, one upstream and one downstream of the propeller. Upstream of the propeller the external pressure \( p_e \) is obtained using Bernoulli:

\[ p_e + \frac{1}{2} \rho V_x^2 = p_0 + \frac{1}{2} \rho V^2. \] (B.10)

The velocity \( V_x \), i.e. the axial velocity on the streamline at a certain \( x \) position, can be expressed as function of the radius of the streamtube at that location using the continuity equation:

\[ V_x = \frac{VA_0}{\pi r^2}. \] (B.11)
substitution leads to:

\[
\int_{r_p}^{r_0} p_e 2\pi r dr = \int_{r_p}^{r_0} \left\{ p_0 + \frac{1}{2} \rho V^2 - \frac{1}{2} \rho \frac{V^2 A_0^2}{\pi^2 r^4} \right\} 2\pi r dr
\]

\[
= p_0 + \frac{1}{2} \rho V^2 (A_0 - A) + \frac{1}{2} \rho V^2 A_0^2 \left( \frac{1}{A_0} - \frac{1}{A} \right)
\]

\[
= p_0 (A_0 - A) + \frac{1}{2} \rho V^2 \left( 2A_0 - \frac{A_0^2}{A} \right)
\]

(B.13)

(B.14)

Equally, downstream of the propeller is holds:

\[
p_e + \frac{1}{2} \rho V_x^2 = p_0 + \frac{1}{2} \rho (V + v_2)^2,
\]

which together with the continuity equation for the downstream region:

\[
V_x = \frac{(V + v_2) A_2}{\pi r^2}
\]

(B.15)

(B.16)

give the pressure integral over the downstream outer part of the control volume:

\[
\int_{r_2}^{r_0} p_e 2\pi r dr = \int_{r_2}^{r_0} \left\{ p_0 + \frac{1}{2} \rho (V + v_2)^2 - \frac{1}{2} \rho \frac{(V + v_2)^2 A_2^2}{\pi^2 r^4} \right\} 2\pi r dr
\]

\[
= \left\{ p_0 + \frac{1}{2} \rho (V + v_2)^2 \right\} (A - A_2) - \frac{1}{2} \rho (V + v_2)^2 A_2^2
\]

\[
= p_0 (A - A_2) + \frac{1}{2} \rho (V + v_2)^2 \left( A - 2A_2 + \frac{A_2^2}{A} \right)
\]

(B.17)

(B.18)

(B.19)

By adding the contribution of the downstream and the upstream part and using the continuity equations

\[(V + v_2) A_2 = V A_0 \text{ and } V A_0 = (V + v_1) A,\]

the equation is rewritten as:

\[
\int_{r_2}^{r_0} p_e 2\pi r dr - p_0 (A_0 - A_2) = \frac{1}{2} \rho v_2 A (v_2 - 2v_1).
\]

(B.20)

The left hand side represents the external force as presented in B.9 while at the right hand side the induced velocities at the propeller plane and in the wake appear. The force of the duct is the external force acting on the control volume. This results in the relation between the duct axial (thrust) force and the velocities in the flow:

\[
T_d = \frac{1}{2} \rho v_2 A (v_2 - 2v_1).
\]

(B.21)

In addition, it is important to remember that for open propeller (without duct) the external force would be zero, such that the right hand side in B.20 is also zero and a relation between the velocities at the propeller plane and in the wake is obtained:

\[
v_2 = 2v_1.
\]

(B.22)

For ducted propeller, it is common to use the thrust ratio \( \tau \) for describing the performance of the system. The thrust ratio is given by the ratio of propeller to total thrust:

\[
\tau = \frac{T_p}{T}.
\]

(B.23)

Generalizing the equation B.22, the velocity in the wake can be expressed as:

\[
v_2 = 2 \tau v_1 - 2(1 - \tau) V,
\]

(B.24)

so the case of open propeller is included when \( \tau = 1 \). By substitution of \( \tau \), the propeller thrust as in B.8 and the duct thrust as in B.21, a quadratic equation for \( v_1 \) is obtained. The root of the equation (after rewriting of the terms containing the total thrust \( T \)) is given by,

\[
\frac{v_1}{V} = -1 + \frac{1}{2\tau} \left\{ 1 + \sqrt{1 + \tau C_T} \right\} \text{ with } C_T = \frac{T}{\frac{1}{2} \rho V^2 \frac{\pi^4}{4} D^2}.
\]

(B.25)
Figure B.2: Actuator disk theory for the ducted propeller. Here the control volume considered is a cylinder with a larger radius $D_3$. This approach proves that the duct force is an external force for the streamtube control volume. Kuiper[3].

where $C_T$ is the thrust load coefficient. This gives directly the total velocity at the propeller plane $v_p = v_1 + V$:

$$\frac{v_p}{V} = \frac{1}{2\tau} \left(1 + \sqrt{1 + \tau C_T}\right).$$

(B.26)

The velocity at the propeller plane increases with smaller $\tau$, when the duct force is larger.

Moreover, the actuator disk theory provides an expression for the efficiency of the system. This is referred to as ideal efficiency, because no viscous nor rotational losses are considered, hence it is an upper limit for the efficiency of the system. The ideal efficiency is given by:

$$\eta_i = \frac{V.T}{V.T + E},$$

(B.27)

where $E$ is the kinetic energy in the slipstream of the propeller:

$$E = \frac{1}{2} \rho (v_1 + V) \pi D_2^2 v_2^2.$$

(B.28)

Substitution of B.26 allows to express the ideal efficiency in terms of the thrust ratio and the thrust load coefficient:

$$\eta_i = \frac{2}{1 + \sqrt{1 + \tau C_T}}.$$

(B.29)

The diagram shown in figure 1.4 of chapter 1 illustrates the change in $\eta_i$ with the propeller loading, parametrized with the thrust ratio. For an equal loading (equal $C_T$), the accelerating duct provide a better efficiency, the decelerating duct a worse efficiency. The differences between open and ducted propellers are larger for large $C_T$, hence fitting an accelerating nozzle around the propeller is more beneficial at high loading.

As reported by Kuiper[3], it has been argue that the derivation of the Actuator disk Theory for ducted propellers cannot follow from the conservation of impulse on the streamtube control volume because the duct force is argued to be an internal force for the control volume. However, it can be proved that this is not the case, and the duct force is an external force, coherently with the previous proof. In order to do so, the control volume in figure B.2 is taken into account, which is a cylinder with a larger diameter $D_3$, surrounding the streamtube. Conservation of mass requires:

$$V(A_3 - A_0) = (V + V_3)(A_3 - A_2),$$

(B.30)
whereas the conservation of momentum for the control volume (following the same procedure to obtain equation B.20) reads:
\[
\int_{r_2}^{r_0} p_e 2\pi r \, dr - p_0 (A_0 - A_2) = (p_3 - p_0)(A_3 - A_2) + \rho (A_3 - A_0) V_3.
\] (B.31)

Furthermore, continuity requires that, when \(A_3 \to 0\) also \(V_3 \to 0\), and for Bernoulli necessarily \(p_3 \to p_0\). Therefore, the equation tends to the zero. The right hand side represents the external forces. When this is zero, it means that the duct force must be an internal force for the outer control volume and therefore it is an external force acting on the inner (streamtube) control volume.

A final remark on the actuator disk theory regards the limitations of the theory. Only axial forces and axial velocities are considered. However, as Gent-Oosterveld explain [38], the duct is subjected to a strong lift force. The lift force relates to the flow circulation induced by the duct. This will largely affect the induced velocities towards the propeller. Figure B.3 provides a sketch of the orientation of the lift force when the inflow has a radial component \(V_r\). The lift force is by definition orthogonal to the inflow \(V\), and it has a major component in radial direction. The radial force is therefore a good indicator of the lift force on the duct, which in terms relates to the axial velocities induced by the duct circulation. A re-formulation of the momentum theory to account for the circulation is discussed by Van Gunsteren[39]. For the sake of this study, it is important to remember that the actuator disk theory gives the most simple model for the ducted propeller, and proves the increase in efficiency that is obtained with a nozzle surrounding the propeller, but effect of viscosity and lift force (hence circulation) are left out.
Propeller and duct geometries

Figure C.1: Marin 19A and 37 duct geometry.
Figure C.2: Ka4-70 propeller geometry, with $D_{HUB}/D = 0.167$. 
Figure C.3: Ka4-70 propeller geometry, with $D_{HB}/D = 0.204$. 
Pressure contours and pressure distributions on the blade and the duct surfaces

Ka4-70 propeller in 19A duct

Figure D.1: $J=0.2$. BEM reference and full RANS solutions.

Figure D.2: $J=0.3$. BEM reference and full RANS solutions.
Figure D.3: $J=0.4$. BEM reference and full RANS solutions.

Figure D.4: $J=0.5$. BEM reference and full RANS solutions.

Figure D.5: $J=0.6$. BEM reference and full RANS solutions.

Figure D.6: $J=0.7$. BEM reference and full RANS solutions.

Figure D.7: $J=0.8$. BEM reference and full RANS solutions.
Figure D.8: J=0.2. Pressure distribution on the blade. Procal reference, RANS-BEM and full RANS solutions. \( r/R = 0.5, 0.7, 0.9 \).

Figure D.9: J=0.3. Pressure distribution on the blade. Procal reference, RANS-BEM and full RANS solutions. \( r/R = 0.5, 0.7, 0.9 \).

Figure D.10: J=0.4. Pressure distribution on the blade. Procal reference, RANS-BEM and full RANS solutions. \( r/R = 0.5, 0.7, 0.9 \).

Figure D.11: J=0.5. Pressure distribution on the blade. Procal reference, RANS-BEM and full RANS solutions. \( r/R = 0.5, 0.7, 0.9 \).

Figure D.12: J=0.6. Pressure distribution on the blade. Procal reference, RANS-BEM and full RANS solutions. \( r/R = 0.5, 0.7, 0.9 \).
Figure D.13: J=0.7. Pressure distribution on the blade. Procal reference, RANS-BEM and full RANS solutions. \( r/R = 0.5, 0.7, 0.9 \).

Figure D.14: J=0.8. Pressure distribution on the blade. Procal reference, RANS-BEM and full RANS solutions. \( r/R = 0.5, 0.7, 0.9 \).

Figure D.15: J=0.2. Angular pressure distribution on the duct. Full RANS solution, experimental results (corrected in phase) and Procal solution with boundary layer displacement thickness at the trailing edge set at \( \delta_{TE}/L_{duct} = 0.04 \). X-planes at \( x/R = 0.25, -0.075, 0.075, -0.075, -0.3 \).
D. Pressure contours and pressure distributions on the blade and the duct surfaces

Figure D.16: J=0.5. Angular pressure distribution on the duct. Full RANS solution, experimental results (corrected in phase) and Procal solution with boundary layer displacement thickness at the trailing edge set at \( \delta^*_T/L_{duct} = 0.04 \). X-planes at \( x/R = 0.25, -0.075, 0.075, -0.075, -0.3 \).

Figure D.17: J=0.2. Duct pressure at 0°, 45°. BEM reference, BEM coupled and full RANS solution.

Figure D.18: J=0.3. Duct pressure at 0°, 45°. BEM reference, BEM coupled and full RANS solution.

Figure D.19: J=0.4. Duct pressure at 0°, 45°. BEM reference, BEM coupled and full RANS solution.

Figure D.20: J=0.5. Duct pressure at 0°, 45°. BEM reference, BEM coupled and full RANS solution.
Figure D.21: J=0.6. Duct pressure at $0^\circ$, $45^\circ$.
BEM reference, BEM coupled and full RANS solution.

Figure D.22: J=0.7. Duct pressure at $0^\circ$, $45^\circ$.
BEM reference, BEM coupled and full RANS solution.

Figure D.23: J=0.8. Duct pressure at $0^\circ$, $45^\circ$.
BEM reference, BEM coupled and full RANS solution.

Figure D.24: Mean pressure distribution on the 19A duct. Reference BEM computation, RANS-BEM distribution and BEM coupled
distribution. Full RANS solution from [11]. J from 0.2 to 0.8.
Ka4-70 propeller in 37 duct

Figure D.25: \( j=0.2 \). BEM reference and full RANS. Ka4-70 in 37 duct. Figure D.26: \( j=0.5 \). BEM reference and full RANS. Ka4-70 in 37 duct.

Figure D.27: \( j=0.2 \). Pressure distribution on the blade. Procal reference, RANS-BEM and full RANS solutions. \( r/R = 0.5, 0.7, 0.9 \). Ka 4-70 in 37 duct.

Figure D.28: \( j=0.5 \). Pressure distribution on the blade. Procal reference, RANS-BEM and full RANS solutions. \( r/R = 0.5, 0.7, 0.9 \). Ka 4-70 in 37 duct.
Figure D.29: $J=0.2$. Duct pressure at $0^\circ$, $40^\circ$.
BEM reference, BEM coupled and full RANS solution.

Figure D.30: $J=0.5$. Duct pressure at $0^\circ$, $40^\circ$.
BEM reference, BEM coupled and full RANS solution.

Figure D.31: Mean pressure distribution on the 37 duct. Reference BEM computation, RANS-BEM distribution and BEM coupled distribution. Full RANS from [14]. $J=0.2$ and $J=0.5$. 
Contour plots

Ka4-70 in 19A duct

Figure E.1: J=0.2. BodyForceX, BodyForceY, Vx/V0 and CPN. RANS-BEM Ka4-70 in 19A duct $D_{HUB}/D = 0.204$.

Figure E.2: J=0.3. BodyForceX, BodyForceY, Vx/V0 and CPN. RANS-BEM Ka4-70 in 19A duct $D_{HUB}/D = 0.204$.

Figure E.3: J=0.4. BodyForceX, BodyForceY, Vx/V0 and CPN. RANS-BEM Ka4-70 in 19A duct $D_{HUB}/D = 0.204$.

Figure E.4: J=0.5. BodyForceX, BodyForceY, Vx/V0 and CPN. RANS-BEM Ka4-70 in 19A duct $D_{HUB}/D = 0.204$. 
Figure E.5: $J=0.6$. BodyForceX, BodyForceY, $V_x/V_0$ and CPN. RANS-BEM Ka4-70 in 19A duct $D_{HUB}/D = 0.204$.  

Figure E.6: $J=0.7$. BodyForceX, BodyForceY, $V_x/V_0$ and CPN. RANS-BEM Ka4-70 in 19A duct $D_{HUB}/D = 0.204$.  

Figure E.7: $J=0.8$. BodyForceX, BodyForceY, $V_x/V_0$ and CPN. RANS-BEM Ka4-70 in 19A duct $D_{HUB}/D = 0.204$.  

Ka4-70 propeller in 37 duct  

Figure E.8: $J=0.2$. BodyForceX, BodyForceY, $V_x/V_0$ and CPN. RANS-BEM Ka4-70 in 37 duct.  

Figure E.9: $J=0.3$. BodyForceX, BodyForceY, $V_x/V_0$ and CPN. RANS-BEM Ka4-70 in 37 duct.
Figure E.10: $J=0.4$. BodyForceX, BodyForceY, $V_x/V_0$ and CPN. RANS-BEM Ka4-70 in 37 duct.

Figure E.11: $J=0.5$. BodyForceX, BodyForceY, $V_x/V_0$ and CPN. RANS-BEM Ka4-70 in 37 duct.

Figure E.12: $J=0.6$. BodyForceX, BodyForceY, $V_x/V_0$ and CPN. RANS-BEM Ka4-70 in 37 duct.

Figure E.13: $J=0.7$. BodyForceX, BodyForceY, $V_x/V_0$ and CPN. RANS-BEM Ka4-70 in 37 duct.

Figure E.14: $J=0.8$. BodyForceX, BodyForceY, $V_x/V_0$ and CPN. RANS-BEM Ka4-70 in 37 duct.
Tip leakage vortex and gap flow analysis

Figure F1: J=0.5 Ka4-70 in 19A duct. Tip leakage vortex detachment angle. Taken from the full RANS computations[11] and the experimental observations [10]. Experimental data refer to cavitation number $\sigma = 1.3$.

Figure F2: J=0.5 Ka4-70 in 19A duct. Tip leakage vortex detachment location. Taken from the full RANS computations[11] and the experimental observations [10]. Experimental data refer to cavitation number $\sigma = 1.3$. 

113
Figure E3: $J=0.2$ Ka4-70 in 19A duct. Tip leakage vortex detachment angle. Taken from the full RANS computations\cite{11} and the experimental observations \cite{10}. Experimental data refer to cavitation number $\sigma = 1.3$.

Figure E4: $J=0.2$ Ka4-70 in 19A duct. Tip leakage vortex detachment location. Taken from the full RANS computations\cite{11} and the experimental observations \cite{10}. Experimental data refer to cavitation number $\sigma = 1.3$. 
Figure E5: J=0.2. Axial velocity contour and streamlines at constant angular positions $\theta = -21.2^\circ$ (blade leading edge), $-10.6^\circ$, $0^\circ$ (mid-chord), $10.6^\circ$, $-21.2^\circ$ (blade trailing edge). Extracted from full RANS solution [11].

Figure E6: J=0.5. Axial velocity contour and streamlines at constant angular positions $\theta = -21.2^\circ$ (blade leading edge), $-10.6^\circ$, $0^\circ$ (mid-chord), $10.6^\circ$, $-21.2^\circ$ (blade trailing edge). Extracted from full RANS solution [11].
Bibliography


[16] L. Kort, Der neue Dusen schraubenartrieb (Werft Reederei Hafen, 1934).


[18] G. Oweis and S. Ceccio, Tip leakage vortex (TLV) variability from a ducted propeller under steady operation and its implication on cavitation inception (Fifth international symposium on cavitation, 2003).


[35] Y. Pan, *A viscous/inviscid interactive approach and its application to wetted or cavitating hydrofoils and propellers with non-zero edge thickness* (University of Texas at Austin, 2009).


