A Rankine Panel Method
as a Tool for the Hydrodynamic Design
of Complex Marine Vehicles

by

DEMETRIOS ALEXIS MANTZARIS

S.M. in Naval Architecture & Marine Engineering,
Massachusetts Institute of Technology (1992)
M.Eng. in Electrical & Electronic Engineering,
Imperial College of Science & Technology, London (1990)

Submitted to the Department of Ocean Engineering
in partial fulfillment of the requirements for the degree of

Doctor of Philosophy in Hydrodynamics

at the

MASSACHUSETTS INSTITUTE OF TECHNOLOGY

February 1998

© Massachusetts Institute of Technology 1998. All rights reserved.
A Rankine Panel Method as a Tool for the Hydrodynamic Design of Complex Marine Vehicles
by
Demetrios Alexis Mantzaris

Submitted to the Department of Ocean Engineering on February 6, 1998, in partial fulfillment of the requirements for the degree of Doctor of Philosophy in Hydrodynamics

Abstract

In the ship designer's quest for producing efficient, functional hull forms, numerical panel methods have emerged as a powerful ally. Their ability to accurately predict forces, motions, and wave patterns of conventional vessels has rendered them invaluable as a design tool. The object of this study was to extend the practical use of such a method to the analysis of non-conventional ships with complex hull geometries.

Features such as twin hulls, transom sterns, and lifting surfaces are all common among today's growing fleet of advanced marine vehicles. Numerical analysis is particularly important for such vessels due to the limited availability of experimental data. Sailing yachts also present a challenge both due to their complex underwater geometries and the small thickness of their sails. Furthermore, the effect of viscosity on the wave flow is worth investigating for any type of ship.

With the above in mind, solutions are presented for the modeling of the free surface flow past lifting bodies, transom sterns, and thin bodies in the context of a linear, time-domain, Rankine panel method. A viscous-inviscid interaction algorithm is also developed, coupling the potential flow method with an integral turbulent boundary-layer model.

Results are presented for conventional ships as well as for two advanced marine vehicles and a sailing yacht, which collectively possess all of the aforementioned geometric complexities. A comparison to experimental data is made whenever possible. For a semi-displacement ship and a catamaran the wave forces, wave patterns, and motions are estimated. In addition, the interaction between the demi-hulls of the catamaran is examined. For the sailing yacht the effect of the lifting appendages on the free surface is investigated, and an approximate, non-linear method is developed to obtain a better evaluation of the steady wave resistance. The significance of correctly modeling the appendages is examined by observing the response amplitude operators for the longitudinal and transverse modes of motion in oblique waves. Finally, a full time-domain simulation of the yacht beating to windward is performed, by simultaneously modeling the flow in air and in water.

Thesis Supervisor: Paul D. Sclavounos
Title: Professor of Naval Architecture
"There is witchery in the sea, its songs and stories, and in the mere sight of a ship... the very creaking of a block... and many are the boys, in every seaport, who are drawn away, as by an almost irresistible attraction, from their work and schools, and hang about the docks and yards of vessels, with a fondness which, it is plain, will have its way."

Richard Henry Dana, Jr.

*Two years before the Mast, 1840*
Acknowledgments

It is difficult to get through any major challenge in life alone, and this would have been particularly true for me during my years at MIT. But I feel very fortunate to have been surrounded by family, friends, and colleagues who made my stay here an enjoyable, productive, and exciting part of my life.

First of all, I would like to express my gratitude to my parents, who have given me all their love and support throughout my education. The person most responsible for where I am today is my father, Lukas, not only for continuously offering me his help and guidance, but perhaps more importantly, for having introduced me to the sport of sailing! I would especially like to thank my mother, Laurel, for being a perfect parent. Besides everything else, she has given me a true bi-cultural upbringing, resulting in an open minded mentality which is essential for progress in life and particularly in engineering.

This thesis is dedicated to my grandmother, Aikaterini, and my aunt Eleni to whom our lengthy separation has been especially hard, as it has been for me. They have always been like parents to me.

On the other hand, this period has given me a chance to be closer to my grandmother Vera, aunt Cheryl and uncle Bill. I have truly cherished the family atmosphere during my regular visits to Chicago.

Many thanks to the rest of my family, Jason, Katherine, Elena, Flora, and Lefteris, who have each contributed in their own special way.

I wish to thank my advisor, Professor Paul Sclavounos, for giving me the opportunity to pursue my dreams and having the patience to put up with me for all these years. I like to believe that some of his high standards and perfectionism have finally rubbed off on me and have helped me improve as a scientist and as a person.

Dave Kring probably deserves this degree more than I do, but since he already has one, I'll just thank him here instead. He was the one person who seemed to suggest a solution to all my research problems, to find a bright spot and to keep me motivated when I thought I had reached a dead end, and the first one to get excited when things were going well.

I would also like to express my appreciation to Professors Nick Patrikalakis and Henrik Schmidt, for serving as members of my thesis committee.

The laboratory for ship and platform flows has been a fun place to do research. Everyone here has been very friendly and helpful but I would particularly like to acknowledge Yonghwan Kim. Had it not been for him, I would still be struggling to find a free computer to finish my runs. Also, Yifeng Huang's help from Houston may have saved me two extra months of work.

I am especially grateful to another member of our research group, Geneviève Tcheou, who during the past two years actually managed to turn me into a more mature person. (Although this doesn't say much, as she would be the first to point out...) She also helped put some balance into my random student life — while doing so, she became my closest friend.

Finally, thanks to Thanos, Babis, Ted, Peter, Thanassis, George, and Hayat for all the wonderful memories. Also to Bill Parcells and the New England Patriots, for providing the standard topic for conversation at lunch, and adding excitement to my weekends.

Financial support has been provided by the Office of Naval Research.
Contents

List of Figures .................................................... xvii
Nomenclature ................................................... xix

I Background ..................................................... 1

1 Introduction .................................................. 3
1.1 Motivation .................................................. 3
1.2 Research History ........................................ 5
1.3 Overview ................................................... 6

2 The Rankine Panel Method .................................. 9
2.1 Mathematical Formulation ............................... 9
  2.1.1 Problem Definition .................................. 9
  2.1.2 Equations of Motion ................................ 11
  2.1.3 Boundary-Value Problem ............................ 11
  2.1.4 Basis Flow ......................................... 12
  2.1.5 Linearization ....................................... 13
  2.1.6 Wave Flow ......................................... 14
2.2 Numerical Implementation .............................. 14

3 Wave Resistance ............................................. 17
3.1 Calm Water Resistance ................................. 17
  3.1.1 Wave Cut Analysis ................................ 19
  3.1.2 Pressure Integration ............................... 19
3.2 Added Resistance due to Waves ....................... 20
II Contributions

4 Viscous Effects

4.1 Introduction ........................................ 25
4.2 The Boundary-Layer Model .......................... 27
  4.2.1 Boundary-Layer Equations ......................... 27
  4.2.2 Boundary Conditions ............................... 29
4.3 Inviscid Flow Compensation for Viscosity .......... 30
  4.3.1 Breathing Velocity ................................ 30
  4.3.2 Body Boundary Condition ......................... 31
  4.3.3 Kinematic Free Surface Boundary Condition .... 32
4.4 The Coupling Algorithm ................................ 33
4.5 Viscous Force Calculations .......................... 34
  4.5.1 Shear Stress Integration ......................... 35
  4.5.2 Conservation of Momentum ......................... 35
4.6 Results ................................................ 36
  4.6.1 Streamline Tracing ................................ 37
  4.6.2 Boundary-Layer Parameters ....................... 37
  4.6.3 Form Factor Calculations ......................... 40
  4.6.4 Wave Patterns and Forces ......................... 42
4.7 Conclusions and Recommendations ................... 43

5 Lifting Surfaces .......................................... 45

5.1 Introduction ........................................ 45
5.2 Formulation .......................................... 46
  5.2.1 Wake Condition ................................. 46
  5.2.2 Basis Flow ....................................... 48
  5.2.3 Wave Flow ........................................ 49
5.3 Numerical Implementation .......................... 49
5.4 Validation ............................................ 50
  5.4.1 Foil in Infinite Flow ............................ 50
## 5.4.2 Surface Piercing Foil
5.5 Conclusions

### 6 Thin Bodies

6.1 Formulation
6.2 Numerical Implementation
6.3 Validation
6.3.1 Foil in Unbounded Fluid
6.3.2 Submerged Horizontal Plate in Heave
6.4 Conclusions

### 7 Deep Transom Sterns

7.1 Formulation
7.2 Numerical Implementation
7.2.1 Transom Conditions
7.2.2 Local Flow
7.3 Validation
7.4 Conclusions

## III Applications

### 8 Conventional Ships

8.1 Series 60
8.2 Model 5415

### 9 Sailing Yachts

9.1 Gridding Issues
9.1.1 Asymmetry
9.1.2 Appendages
9.2 The Steady Resistance Problem
9.2.1 A Non-Linear Extension
9.2.2 Appendage–Free Surface Interaction .......................... 96
9.3 Free Motion Simulations in Waves ................................ 97
  9.3.1 Head Seas .................................................. 97
  9.3.2 Appendage Modeling ........................................ 98
9.4 Coupling of Aerodynamic and Hydrodynamic Flows ............. 100

10 Advanced Marine Vehicles ........................................ 105
  10.1 Semi-Displacement Ship ...................................... 105
    10.1.1 Steady Wave Resistance .................................. 106
    10.1.2 Motions in Head Seas ................................... 107
  10.2 Catamaran .................................................... 108
    10.2.1 Lewis Form Catamaran .................................. 109
    10.2.2 Demi-hull Interaction ................................... 111
    10.2.3 Wave Patterns ........................................... 112
    10.2.4 Non-linear Motions ...................................... 113

Concluding Comments ................................................ 117

A The Boundary Layer Model ........................................ 119
  A.1 Definition of Main Parameters ................................. 119
  A.2 Flow over Solid Surfaces .................................... 120
  A.3 Wake Flows .................................................. 122
  A.4 Boundary Conditions ......................................... 123

Bibliography ......................................................... 124
List of Figures

2-1 A graphic representation of the problem ........................................ 10

4-1 A flowchart of the coupling algorithm ........................................ 33

4-2 Control volume used for the viscous drag calculation .................... 36

4-3 Streamlines traced on an IACC hull (not to scale) ........................ 37

4-4 Velocity distribution on two streamlines .................................... 38

4-5 Displacement thickness on two streamlines .................................. 38

4-6 Shape parameter on two streamlines .......................................... 39

4-7 Breathing velocity on two streamlines ....................................... 39

4-8 Skin friction coefficient on two streamlines ................................ 40

4-9 Form factor variation with Froude number for an IACC hull .......... 41

4-10 The effect of the viscous boundary-layer on wave elevation (a) and resistance (b) ................................................................. 42

5-1 The pressure distribution of a two-dimensional Karman-Trefftz section with parameters $x_c = 0.1$, $y_c = 0.1$, $\tau = 10$, at a $2^\circ$ angle of attack .............................................................. 52

5-2 The convergence of the lift (a) and drag (b) coefficients for an elliptical foil with aspect ratio $A = 10.19$ and the section of figure 5-1, as computed by pressure and Trefftz plane integration. The analytic estimation is made by correcting the two-dimensional lift and drag coefficients along the foil using Prandtl's lifting line theory ........................................... 53

5-3 Demonstration of good numerical behavior at the tip of an elliptical foil with $A = 10.19$. Shown, are the spanwise distribution of bound vorticity (a), and the chordwise distribution of potential at several sections of the foil (b). The section used is as described in figure 5-1 53
5-4 Lift (a) and drag (b) sensitivity to the position of the wake. $\theta = 0$ implies that the wake is aligned with the free stream. The foil used has an elliptical planform with $A = 10.19$ and the section shown in figure 5–1.

5-5 Wave patterns of a surface piercing foil at $F_n = 0.3$ and $F_n = 1.0$.

5-6 The lift coefficient (a) and the jump in wave elevation at the trailing edge (b) as a function of Froude number, for a surface piercing foil.

5-7 The wave elevation at the pressure and suction side of a surface piercing foil and its wake, for a range of speeds.

6-1 The lift coefficient of a rectangular planform foil with $A = 0.5$ as a function of thickness-to-chordlength ratio (a), and the distribution of the potential jump over the chordlength of the plate at mid-span (b).

6-2 Heaving circular plate. (a) The convergence of the heave added mass in an unbounded fluid and comparison with theory. (b) The wave elevation above the plate heaving with amplitude $A$, one radius $(d/2)$ below the free surface. The frequency of heave is $\omega = 1.5(g/d)^{1/2}$.

6-3 The heave added mass (a) and damping (b) of a horizontal circular plate submerged at a depth of one radius. The damping is calculated from both the radiation and diffraction problems.

7-1 Profile view of a typical deep transom stern.

7-2 The lines drawing of the test transom-stern ship used in this study. Only the underwater portion of the hull is shown.

7-3 Contour plots of the wave elevation and pressure distribution on a transom-stern hull, for an increasing number of panels.

7-4 A comparison between the present time-domain method (a) and an equivalent frequency-domain method (b) for the steady flow solution past a transom-stern hull.
7-5 The heave (a) and pitch (b) RAO of a transom-stern ship. Convergence with an increasing number of panels and comparison with a frequency-domain method. ............................... 75

8-1 The lines drawing of the Series 60. ........................................ 82
8-2 Domain sensitivity for the Series 60. ........................................ 82
8-3 Contour plots for the wave elevation and pressure distribution on the Series 60 hull, at Froude number 0.316, for an increasing number of panels. ........................................ 83
8-4 The wave profile along the hull (a) and the wave heights at a cut along y/L=0.108 (b) for the Series 60 hull at Froude number 0.316. Results are shown for an increasing number of panels and are compared to experiments. ........................................ 84
8-5 The lines drawing of Model 5415. ........................................ 85
8-6 The wave profile along the hull (a) and the wave heights at a cut along y/L=0.097 (b) for Model 5415 at Froude number 0.2755. Results are shown for an increasing number of panels and are compared to experiments. ........................................ 85
8-7 The wave profile along the hull (a) and the wave heights at a cut along y/L=0.097 (b) for Model 5415 at Froude number 0.4136. Results are shown for an increasing number of panels and are compared to experiments. ........................................ 86

9-1 The lines drawing of an IACC yacht. ........................................ 90
9-2 A comparison of polar and rectangular free surface gridding. ....... 92
9-3 The discretization of the hull and keel of an IACC yacht. .......... 93
9-4 The resistance coefficient of an IACC yacht. Convergence of the numerical results using pressure integration together with the waterline integral (a) and using simple pressure integration (b). The results by the approximate non-linear method are also shown in (b). The coefficient is based on the surface of the hull below the z = 0 plane. .... 94
9-5 Contour plots of the wave elevation and body pressure for an IACC yacht with and without the keel and bulb in place. .......................... 96

9-6 The Heave (a) and Pitch (b) RAO for an IACC hull traveling at Froude number 0.347 in head waves. ................................. 97

9-7 The oblique wave heave (a), pitch (b), roll (c), and yaw (d) RAO for an IACC yacht under heel and leeway. Results are presented for no fin-keel, for the hull with the fin-keel modeled using lifting line theory, and for the hull with the fin-keel paneled and treated according to chapter 5. 99

9-8 A full simulation of an IACC yacht beating to windward in waves. ... 101

9-9 The effect of the sails on the oblique wave heave (a), pitch (b), roll (c), and yaw (d) RAO for an IACC yacht under heel and leeway. The apparent wind velocity is 20 knots from a direction of 30° off the direction of motion. The yacht is traveling at a speed of 9 knots, through waves incident at 45 degrees from the beam. ............................. 102

10-1 The calm-water wave pattern of the TGC770 FastShip at 40 knots. ... 106

10-2 Residuary power estimates for the TGC770 FastShip. ................ 107

10-3 Comparison of RAO values for heave (a) and pitch (b) for the TGC770 FastShip. SWAN 1 is the frequency-domain method, SWAN 2 is the time-domain method, and the experiments are from Branner and Sangberg. ................................. 108

10-4 Heave–heave added mass and damping coefficient predictions compared to experiment for a Lewis form catamaran at $F_n = 0.3$ .......................... 109

10-5 Heave–pitch added mass and damping coefficient predictions compared to experiment for a Lewis form catamaran at $F_n = 0.3$ .......................... 110

10-6 Pitch–pitch added mass and damping coefficient predictions compared to experiment for a Lewis form catamaran at $F_n = 0.3$ .......................... 110

10-7 Wave resistance coefficient as a function of speed for a high-speed catamaran with various demi-hull separation ratios. ......................... 111

10-8 Roll RAO for a catamaran at $F_n = 0.8$ for two separation ratios. ... 112
10-9 Heave and pitch RAOs for a catamaran at various separation ratios and speeds. ........................................ 113

10-10 Steady wave patterns for a catamaran from moderate to high speeds. ........................................ 114

10-11 Snapshots of the submerged hull surface for a catamaran at $F_n = 0.3$ in a regular head sea; and the corresponding time record for the linear and non-linear heave motions. ............................ 115
Nomenclature

\( A \) aspect ratio

\( C \) matrix of hydrostatic restoring coefficients

\( C_E \) entrainment coefficient

\( C_D \) induced drag coefficient

\( C_d \) intersection of \( S_\infty \) with \( z = 0 \) plane

\( C_f \) frictional coefficient

\( C_L \) lift coefficient

\( C_W \) wave-making resistance coefficient

\( \mathcal{F} \) Fourier transform of wave elevation

\( F_D \) total hydrodynamic force

\( F_R \) Froude number

\( G \) Rankine source potential

\( g \) acceleration of gravity

\( H \) shape parameter

\( \mathcal{H} \) free wave spectrum of vessel

\( L \) length of waterline

\( M \) inertial matrix of body

\( m_j \) m-term in the \( j^{th} \) direction

\( \mathbf{n} = (n_1, n_2, n_3) \) unit normal vector

\( p \) pressure

\( R_w \) wave resistance

\( R_x \) Reynolds number based on length \( x \)

\( S_B \) exact wetted surface of body

\( S_\beta \) linearized wetted surface of body

\( S_B^* \) image of \( S_B \) about \( z = 0 \) plane

\( S_d \) portion of \( S_\infty \) below \( z = 0 \) plane

\( S_F \) exact description free surface
\[ S_p \] free surface in undisturbed position
\[ S_p \] surface representing "thin" body (plate)
\[ S_W \] surface representing vortex wake
\[ S_\infty \] semi-infinite vertical plane below free surface, far downstream
\[ s \] span
\[ T \] (subscript) trailing edge of lifting surface or transom
\[ U \] magnitude of mean velocity
\[ V_e \] breathing velocity
\[ \bar{W} \] mean velocity of body
\[ x = x_1 \] coordinate in direction of mean body motion
\[ y = x_2 \] horizontal coordinate normal to direction of body motion
\[ z = x_3 \] vertical coordinate
\[ \alpha \] angle of attack
\[ \gamma \] flare angle of hull
\[ \Delta \] "jump" operator on dipole sheet
\[ \delta^* \] displacement thickness
\[ \bar{\delta} \] displacement of body about mean position
\[ \epsilon \] hull slenderness parameter
\[ \zeta \] free surface wave elevation
\[ \theta \] momentum thickness
\[ \kappa_x \] wavenumber of wave propagating in x-direction
\[ \kappa_y \] wavenumber of wave propagating in y-direction
\[ \nu \] kinematic viscosity
\[ \xi_R = (\xi_4, \xi_5, \xi_6) \] rigid body rotation
\[ \xi_T = (\xi_1, \xi_2, \xi_3) \] rigid body translation
\[ \rho \] density
\[ \sigma \] strength of source distribution
\[ \Phi \] disturbance potential of basis flow
\[ \varphi \] perturbation flow velocity potential
\[ \Psi \] total disturbance velocity potential
Part I

Background

The schooner yacht America (1851)
Chapter 1

Introduction

1.1 Motivation

A ship differs from any other large engineering structure in that — in addition to all its other functions — it must be designed to move efficiently through the water with a minimum of external assistance. Another requirement besides good smooth-water performance, is that under average service conditions at sea, the ship shall not suffer from excessive motions, wetness of decks, or lose more speed than necessary in bad weather.

Although ships have been traversing the oceans for millennia, there has not always been a systematic way of satisfying the above criteria. Designers have relied on centuries of tradition and their own experience and intuition, but in order to actually ensure the desirable ship performance, it is necessary to possess knowledge of the hydrodynamics of the hull and the propulsion system.

Because of the complicated nature of ship hydrodynamics, early recourse was made to experiments for its understanding. But even after model testing was revolutionized by Froude in the 1860’s, it still remains a very time-consuming and expensive process.

Ever since the advent of the digital computer, numerical methods have been gaining popularity as an alternative to towing tank testing. The field has been rapidly evolving over the past 15 years, ultimately leading to the development of fully three-
dimensional boundary integral element methods.

Such methods, also known as panel methods, discretize boundaries of the fluid into elements with an associated singularity strength, impose appropriate boundary conditions, and most use linear potential flow theory to attempt to numerically reproduce the flow past the ship. A class of such methods, which has produced especially promising results, employs the Rankine source as the elementary singularity. It is very flexible in the free surface conditions that it can enforce, and when combined with a time-domain approach, can even be extended to include non-linear effects.

There are, however, several limitations of these methods that currently prevent them from totally replacing the experiment as a primary means of evaluating a design.

One such restriction is that of geometric complexity. Present linear methods have not had success in treating ships with deep transom sterns, for example. Also, agreement with experiments has been questionable for hulls with significant flare, or with overhang at the stern. Both the above characteristics are very common in today’s fleet of commercial ships and sailing yachts, and must be treated properly, using a sound theoretical basis, while overcoming any numerical difficulties.

Hulls with thin appendages compared to the overall dimensions of the ship, are another geometric complexity presenting a challenge for panel methods. Sails, offshore platform damping plates, and even rudders are a few such examples, where an inordinate amount of panels are required to overcome the numerical difficulties associated with the proximity of the surfaces on which elementary singularities are distributed.

Another limitation is the fact that panel methods do not account for viscosity. The interaction between the wave flow and the viscous boundary layer cannot be captured with existing potential flow methods. This interaction is, however, inherently present in towing tank tests$^1$.

Many types of advanced marine vehicles such as catamarans, SWATH, and SES,

---

$^1$Even in towing tank tests, however, this interaction might not be accurately evaluated due to scaling effects. In addition, such tests rely on the rather crude Froude hypothesis to determine the residuary resistance. Therefore, a numerical method that accounts for viscous effects has the potential of being more accurate than model tests.
operate with their hulls producing a significant amount of lift. In addition, appendages such as keels, rudders, and winglets are vital for the operation of sailing yachts. Therefore, the treatment of circulation is another feature that would enormously help in establishing panel methods as a design tool, especially since the amount of experimental data available for non-conventional ships is limited.

This thesis will address all of the above issues, attempting to bring a Rankine panel method closer to being a ship designer's primary tool for the hydrodynamic evaluation of complex marine vehicles.

1.2 Research History

Boundary integral element methods form the basis of the majority of the computational algorithms for the numerical solution of the forces and wave patterns of bodies traveling near the boundary between two fluids. In 1976-77, the work of Gadd [9] and Dawson [7] ignited a class of such methods which use the Rankine source as their elementary singularity. Known as Rankine panel methods, they distribute these singularities on the discretized free surface, as well as on the body, and solve for their unknown strengths.

The advantage of such methods is the freedom to impose a wide range of free surface boundary conditions. This leads to a flexibility of linearization about a basis flow or even extension to the fully non-linear problem. In 1988, Sclavounos and Nakos [31] presented an analysis for the propagation of gravity waves on a discrete free surface which instilled confidence that such a method could faithfully represent ship forces and wave patterns, despite the distortion of the wave system introduced due to the free surface discretization. Their work led to the development of a frequency domain panel method [30], dubbed SWAN (Ship Wave ANalysis), capable of accurately predicting the flow, as reported for several applications [32, 47].

The time domain formulation, and simultaneous solution of the equations of motion with the wave flow, is another feature which allows for the future inclusion of non-linearities in the unsteady problem. Kring [18, 20] extended the work of Nakos
and Sclavounos to the time domain, preserving their methodology, thus taking advantage of the experience gained by the evolution of SWAN and ensuring that the underlying numerical method faithfully represented the problems posed.

Such linear methods give reliable estimates of the motions, wave patterns, and forces for most practical hull forms. But for certain extreme cases, such as ships with large curvature or slope at the waterline, the linearized free surface conditions become inconsistent. Recently, fully non-linear panel methods have been developed, which remove the inconsistencies inherent in the linearization process. At present, however, these methods are either not general enough, or too inefficient to be routinely used for evaluating the performance of real applications. Xia [50], Ni [36], Jensen [15, 16], and Raven [40] have developed methods which deal only with the problem of steady motion. The transient method of Beck and Cao [1, 5], is more flexible but very computationally intensive.

This study will be concerned with extending a linear panel method to include viscous effects, lifting surfaces, and treatment of thin bodies and complex geometries such as transom sterns. Thus, it will immediately become a practical tool of hydrodynamic design. The time-domain, boundary integral element approach will ensure that as computer power increases in the future, the method will be readily extendible to non-linear computations. In fact, some non-linear extensions have already been incorporated. There is currently capability to include the effect of non-linearities such as systems of active control and viscous roll damping [45], while the recent work of Huang [13] based on a weak scatterer hypothesis has led to significant improvements in the non-linear seakeeping problem.

1.3 Overview

As mentioned above, the present thesis will extend a Rankine panel method to include viscous effects, and provide a means of treating lifting surfaces, thin bodies, and the flow past transom sterns. These seemingly unrelated topics have as a common goal to enable the ship designer to analyze the flow past complex hulls such as semi-
1.3. Overview

displacement ships, catamarans, and sailing yachts.

*Part I* provides the background theory and methodology which are necessary for the extensions of the Rankine panel method, and consists of two chapters. Chapter 2 gives an overview of the basic time-domain Rankine panel method and chapter 3 reviews the method used for the calculation of the steady wave resistance and the added resistance due to waves.

*Part II* presents the new contributions of this work. Chapter 4 extends the method to include viscosity effects and their interaction with the wave flow. A direct viscous-inviscid interaction algorithm is developed using the Rankine panel method and an integral turbulent boundary-layer method. Chapter 5 gives a method to treat free surface flows with lift, by modeling the trailing vortex wake and employing an appropriate Kutta condition. Chapter 6 provides an extension for bodies with infinitesimally small thickness by modeling them as dipole sheets. Finally, chapter 7 presents a solution for the numerical treatment of flows past deep transom sterns.

*Part III* presents some case studies, in which the above extensions are utilized to assess the performance of various advanced marine vehicles. In chapter 8, the method is used with two conventional ships and is tested for convergence and for agreement with experiments. Chapter 9 presents an extensive study of a sailing yacht, including the aerodynamic forces and a full time-domain simulation of the vessel under sail. Chapter 10 examines the forces, wave patterns and motions of a semi-displacement ship and a catamaran.
CHAPTER 2

THE RANKINE PANEL METHOD

This chapter will give an overview of a time domain, linear Rankine panel method, with the purpose of providing a framework on which the extensions of the following chapters will be based. The problem is formulated in section 2.1, and the numerical implementation in an efficient algorithm is given in section 2.2. For a more detailed description, the reader may refer to the Ph.D. thesis of Kring [18].

2.1 Mathematical Formulation

2.1.1 Problem Definition

Figure 2-1 displays a vessel with wetted surface $S_B$, interacting with the surface of the sea, $S_F$. It may have a mean forward speed, $\vec{W}$, and a reference frame $(x, y, z)$ is fixed to this steady motion. All quantities below are taken with respect to this frame of reference.

The body may also perform time-dependent motions about this frame of reference in the six rigid-body degrees of freedom. Its displacement $\delta$, at position $\vec{x} = (x, y, z)$, may then be written as

$$\delta(\vec{x}, t) = \bar{\xi}_T(t) + \bar{\xi}_R(t) \times \vec{x}$$  \hspace{1cm} (2.1)

where \(\bar{\xi}_T = (\xi_1, \xi_2, \xi_3)\) is the rigid body translation and \(\bar{\xi}_R = (\xi_4, \xi_5, \xi_6)\) is the rigid
Figure 2-1: A graphic representation of the problem
2.1. Mathematical Formulation

2.1.2 Equations of Motion

A direct application of Newton’s Law leads to the equations of motion of the vessel.

\[ M \dddot{\xi}(t) = \ddot{F}_D(\xi, \dot{\xi}, \ddot{\xi}, t) - C(\ddot{\xi}(t)) \] (2.2)

\( M \) above is the inertial matrix for the body and \( C \) is the matrix of hydrostatic restoring coefficients. In order to obtain the hydrodynamic forces \( \ddot{F}_D \), a potential flow boundary-value problem is solved.

2.1.3 Boundary-Value Problem

It is assumed that inertial and gravity forces will dominate wave propagation and therefore the flow within the fluid domain is inviscid, irrotational, and incompressible. Under this assumption, the flow is governed by a total disturbance velocity potential \( \Psi(x, t) \), which satisfies the Laplace equation in the fluid domain and is subject to the kinematic and dynamic free surface conditions

\[ \left[ \frac{\partial}{\partial t} + (\nabla \Psi - \ddot{W}) \cdot \nabla \right] [z - \zeta] = 0 \] (2.3)

\[ \left( \frac{\partial}{\partial t} - \ddot{W} \cdot \nabla \right) \Psi + g \zeta + \frac{1}{2} \nabla \Psi \cdot \nabla \Psi = 0 \] (2.4)

which are imposed at the instantaneous position of the free surface, \( \zeta(x, y, t) \). Here, the free surface has been assumed to be single-valued, thereby neglecting non-linear effects such as breaking waves and spray.

The no-flux body boundary condition imposed on the wetted surface of the hull is given by

\[ \frac{\partial \Psi}{\partial n} = (\ddot{W} + \frac{\partial \ddot{x}}{\partial t}) \cdot \vec{n} \] (2.5)

To close the exact problem, initial conditions are posed for \( \frac{\partial \Psi}{\partial t}(\ddot{x}, t), \Psi(\ddot{x}, t), \) and the body displacement and velocity. The gradients of the disturbance potential are
also required to vanish at a sufficiently large distance from the vessel at any given finite time.

2.1.4 Basis Flow

The flow is linearized about a dominant basis flow. There are two linearizations that are commonly applied to this three-dimensional problem. One is the classical Neumann–Kelvin linearization, where a uniform stream is taken as the basis flow. For most realistic hull forms, however, the best results are obtained by linearizing about a double-body flow, as first proposed by Gadd [9] and Dawson [7].

There is a choice of methods for the solution of the double-body flow using a panel method. One such method consists of distributing sources on the body boundary $S_B$, and its mirror image about the $z = 0$ plane $S_{B*}$, and then using the boundary condition (2.5) to derive an integral equation for their unknown strengths, $\sigma(\vec{x})$.

$$\int \int_{S_B \cup S_{B*}} \sigma(\vec{x}') \frac{\partial G(\vec{x}'; \vec{x})}{\partial n} \, d\vec{x}' = \vec{W} \cdot \vec{n}$$

(2.6)

where $G(\vec{x}'; \vec{x}) = \frac{1}{|\vec{x}'-\vec{x}|}$ is the Rankine source potential, and $\vec{x} \epsilon S_B$

Note, however, that if the double body flow involves circulation, then a solution cannot be found in terms of a pure Rankine source distribution on the body. In this case it is necessary to use either dipoles or vortices in addition to sources. The application of Green's second identity, together with the body boundary condition (2.5), leads to the potential formulation of the double-body, which may be expressed as follows

$$2\pi \Phi(\vec{x}) - \int \int_{S_B \cup S_{B*}} (\vec{W} \cdot \vec{n}) \, G(\vec{x}'; \vec{x}) \, d\vec{x}' +$$

$$\int \int_{S_B \cup S_{B*}} \Phi(\vec{x}') \frac{\partial G(\vec{x}'; \vec{x})}{\partial n} \, d\vec{x}' = 0$$

(2.7)

where $\Phi$ is the disturbance potential of the double-body flow and $x \epsilon S_B$.

This method will be extended in section 5.2.2 for the case of a flow with circulation.
2.1.5 Linearization

Free Surface Boundary Conditions

Assuming that the total disturbance velocity potential, $\Psi$, consists of a dominant basis flow component $\Phi$, and a perturbation correction $\varphi$, the kinematic and dynamic free surface conditions may be linearized and applied at $z = 0$ as follows:

\[ \frac{\partial \zeta}{\partial t} - (\bar{W} - \nabla \Phi) \cdot \nabla \zeta = \frac{\partial^2 \Phi}{\partial z^2} \zeta + \frac{\partial \varphi}{\partial z} \]  
(2.8)\]

\[ \frac{\partial \varphi}{\partial t} - (\bar{W} - \nabla \Phi) \cdot \nabla \varphi = -g \zeta + [\bar{W} \cdot \nabla \Phi - \frac{1}{2} \nabla \Phi \cdot \nabla \Phi] \]  
(2.9)\]

A further decomposition of the perturbation potential into instantaneous and memory components is used to obtain a numerically stable scheme for the integration of the equations of motion, as discussed by Kring [18].

Body Boundary Conditions

Linear theory allows the decomposition of the wave disturbance into independent incident, radiated and diffracted components. As first shown by Timman and Newman [48], the body boundary condition of the radiation component linearized about the mean position of the hull, takes the following form

\[ \frac{\partial \varphi}{\partial n} = \sum_{j=1}^{6} \left( \frac{d \xi_j}{dt} + \xi_j m_j \right) \]  
(2.10)\]

with,

\[ (n_1, n_2, n_3) = \vec{n} \]

\[ (n_4, n_5, n_6) = \vec{x} \times \vec{n} \]

\[ (m_1, m_2, m_3) = (\vec{n} \cdot \nabla)(\bar{W} - \nabla \Phi) \]

\[ (m_4, m_5, m_6) = (\vec{n} \cdot \nabla)(\vec{x} \times (\bar{W} - \nabla \Phi)) \]  
(2.11)\]
The m-terms, $m_j$, provide a coupling between the steady basis flow and unsteady body motion.

The diffraction body boundary condition states that the normal velocity of the sum of the incident and diffraction velocity potentials equals zero on the mean position of the hull.

### 2.1.6 Wave Flow

The Laplace equation is enforced in the fluid domain by a distribution of Rankine sources and dipoles over the free surface and hull. Application of Green's second identity leads to a boundary integral formulation for the perturbation potential.

$$2\pi \varphi(\vec{x}) - \int_{S_p \cup S_b} \frac{\partial \varphi(\vec{x}')}{\partial n} G(\vec{x}; \vec{x}') \, dx' + \int_{S_p \cup S_b} \varphi(\vec{x}') \frac{\partial G(\vec{x}'; \vec{x})}{\partial n} \, dx' = 0 \quad (2.12)$$

where $G(\vec{x}'; \vec{x}) = \frac{1}{|\vec{x} - \vec{x}'|}$ is the Rankine source potential, $S_{\bar{F}}$ is the undisturbed position of the free surface, $S_{\bar{B}}$ is the wetted surface of the mean position of the stationary hull in calm water, and $\bar{x} \epsilon (S_{\bar{F}} \cup S_{\bar{B}})$.

### 2.2 Numerical Implementation

The three unknowns in the above formulation are the velocity potential $\varphi$, the wave elevation $\zeta$, and the normal velocity $\varphi_n$. To solve for these unknowns, the free surface conditions (2.8) and (2.9), which form a pair of evolution equations, and the integral equation (2.12) are satisfied numerically by a time-domain Rankine panel method.

The Rankine panel method discretizes the hull surface and a portion of the $z = 0$ plane representing the free surface. Each of the unknowns is approximated independently by a set of bi-quadratic spline functions that provide continuity of value and of first derivative across panels.

The evolution equations employ an explicit Euler integration to satisfy the kine-
matic free surface condition and an implicit Euler integration to satisfy the dynamic free surface boundary condition.

A numerical, wave-absorbing beach is used to satisfy the radiation condition, since only a finite portion of the free surface is considered by the panel method.

Thus, a solution for the wave flow is produced and the equations of motion (2.2) are integrated at each time step in order to satisfy the radiation body boundary conditions.

A more detailed discussion of the formulation, numerical method, and applications can be found in the work of Kring [18], Nakos and Sclavounos [31], and Sclavounos et al.[45, 44]
CHAPTER 3

WAVE RESISTANCE

One of the most practical applications of panel methods, is the prediction of the wave forces after the flow field has been solved.

In light of the three-dimensional Rankine panel method considered, this chapter will review a consistent definition of wave resistance for the linearized problem, and will introduce means of calculating the added resistance due to waves.\textsuperscript{1}

3.1 Calm Water Resistance

The wave-making resistance of a ship is the net fore-and-aft force upon the ship due to the fluid pressure acting normally on all parts of the hull. This pressure may be readily obtained from Bernoulli's equation, after having determined the potential flow from the solution of the boundary value problem which has been formulated in the preceding chapter.

Due to the linearization of the problem about the calm water surface, it is necessary to decompose the wetted surface into the portion $S_B$ which lies beneath $z = 0$, and an extra surface $\delta S_B$, which accounts for the difference between the exact wetted surface and $S_B$. The integration of the pressure over $\delta S_B$, often referred to as the run-

\textsuperscript{1}The panel method on which this work has been based was actually modified to perform the wave and added resistance calculations. It was not felt, however, that this constituted an original contribution and is thus discussed here as part of the background theory and methodology.
up, may be collapsed into a line integral. This integral is of magnitude proportional to the square of the wave elevation, and is therefore inconsistent with the linearization of the free surface conditions (2.8), (2.9) which omit terms of comparable order.

For surface-piercing bodies, the only consistent definition of the linearized wave resistance by pressure integration is the one adopted by thin-ship theory, where the assumption of geometrical slenderness is employed in order not only to drop the quadratic pressure terms of the perturbation flow, but also to linearize the hull thickness effect by collapsing the body boundary condition on the hull centerplane.

On the other hand, for full shaped vessels, the quadratic terms of the perturbation flow in Bernoulli's equation and in the run-up are considerable and their omission is found to cause over-prediction of wave resistance, in a fashion similar to the performance of thin-ship theory.

The solution is given by Nakos and Sclavounos [33], who show that the only consistent definition of the wave resistance with the linearization of the problem follows from conservation of momentum and not from pressure integration.

The wave resistance, as defined above, may be calculated by applying the momentum theorem to a control volume bounded by the exact wetted surface of the hull, by the exact position of the free surface, and by a closing surface at infinity. By virtue of the radiation condition, the closing surface may be replaced by a vertical plane, \( S_\infty \), normal to the ship axis at a large distance downstream. Taking into consideration the asymptotically small magnitude of the wave disturbance in the far-field, it follows that

\[
R_w = -\frac{\rho g}{2} \int_{C_d} \zeta^2 \, dy - \frac{\rho}{2} \int_{S_d} \left[ -\frac{\partial \varphi^2}{\partial x} + \frac{\partial \varphi^2}{\partial y} + \frac{\partial \varphi^2}{\partial z} \right] \, dS
\]

(3.1)

where \( C_d \) is the intersection of \( S_\infty \) with the \( z = 0 \) plane, and \( S_d \) is the part of \( S_\infty \) lying below \( z = 0 \).

Equation (3.1) may be evaluated either in terms of far-field quantities through wave-cut analysis, or in terms of near-field quantities through pressure integration.
3.1. Wave Cut Analysis

The principal properties of the transverse wave cut method are briefly reviewed below. Further details may be found in Eggers et al [8].

Consider a cut of the wave pattern perpendicular to the steady track of the vessel for \(-\infty < y < +\infty\). The transverse Fourier Transform of the wave elevation, \(\zeta(x, y)\) and its x-derivative are given by (3.2) and (3.3) respectively.

\[
\mathcal{F}(x, \kappa_y) = \int_{-\infty}^{+\infty} \zeta(x, y) e^{ik_y y} dy 
\]

(3.2)

\[
\mathcal{F}_x(x, \kappa_y) = \int_{-\infty}^{+\infty} \frac{\partial \zeta(x, y)}{\partial x} e^{ik_y y} dy 
\]

(3.3)

In the limit as \(x \to \infty\) the vessel’s free wave spectrum is defined as

\[
\mathcal{H}(\kappa_y; x) = 2 \left[ \mathcal{F}(x, \kappa_y) + i \frac{\mathcal{F}_x(x, \kappa_y)}{\kappa_x} \right] e^{ik_x x} \kappa_y e^{-\infty, +\infty} 
\]

(3.4)

where the wavenumbers, \(\kappa_x\) and \(\kappa_y\), are normalized by \(g/U^2\) and satisfy the dispersion relation, \(\kappa_x^2 = \sqrt{\kappa_x^2 + \kappa_y^2}\), by virtue of the Kelvin condition which is asymptotically valid in the far-field.

The wave resistance in terms of far-field quantities, is therefore given by

\[
R_w = \frac{\rho U^2}{8\pi} \int_0^{\infty} |\mathcal{H}(\kappa_y)|^2 \frac{\sqrt{1 + 4\kappa_y^2}}{1 + \sqrt{1 + 4\kappa_y^2}} d\kappa_y 
\]

(3.5)

3.1.2 Pressure Integration

An equivalent evaluation of (3.1) may be performed in terms of near-field quantities. The momentum theorem is applied to the fluid enclosed by the linearized free surface, the body, and the closing surface at infinity. In this case there is a momentum flux across the linearized free surface, which may be evaluated in terms of the wave elevation using the Neumann–Kelvin free surface conditions. An application of Stoke’s theorem to the integral over \(S_P\), and substitution into (3.1) then leads to an equivalent
expression for the wave resistance using pressure integration.

\[
R_w = \int \int_{S_{\alpha}} p n_{1} dS - \frac{\rho g}{2} \int_{w_{1}} \frac{\zeta^2 n_{1}}{\cos \gamma} dl
\]

(3.6)

where \( \gamma \) is the flare angle of the hull.

### 3.2 Added Resistance due to Waves

The added resistance of a ship in regular waves is defined as the mean value of her total resistance minus her calm water resistance. Linear theory allows the independent solution of the steady and unsteady components of the flow. In regular waves this simplifies the definition of the added resistance as the mean value of the component of resistance which is proportional to the square of the wave amplitude. This section will present the principles of modeling and computation of the added resistance, starting from a pressure integration over the wetted surface of the hull.

Nakos [30] has suggested that, for the steady flow, the linearization is to be performed assuming that the product of some slenderness parameter times the square of the Froude number \( (\epsilon F_{n}^2) \) is small. As seen in the previous section, this leads to an inconsistency in the definition of steady wave resistance in terms of pressure integration.

The question arises on whether similar problems are encountered in the evaluation of the added resistance. For the unsteady flow, however, the linearization is based on the assumption of a small ambient wave amplitude. When calculating added resistance, the quantity \( \epsilon F_{n}^2 \) is assumed \( O(1) \), thus removing any inconsistency associated with the hull thickness effect.

Sclavounos and Nakos [46] have presented a model for the computation of added resistance using a frequency-domain Rankine panel method. The model used with the present time-domain method is essentially the same, and will only be briefly outlined here.

The total resistance of the ship may be found by pressure integration over the
wetted surface of the hull, $S_B$. For compatibility with the discretization scheme, the wetted surface of the hull is divided into the wetted surface of the vessel at rest in calm water $S_B$ plus an additional surface $\delta S_B$, to account for the motion of the hull and the shape of the dynamic waterline. Hence,

$$R_w = \int \int_{S_B} p \, n_1 \, dS + \int \int_{\delta S_B} p \, n_1 \, dS$$

(3.7)

the first term above may be evaluated by a direct pressure integration on the mean surface of the hull, whereas the second term is calculated in terms of a waterline integral.

The pressure, normal vector, and $\delta S_B$ in (3.7) are required at the instantaneous position of the hull. These may be easily expressed in terms of the values at the mean position of the hull, using a Taylor series expansion.

It is assumed that the unsteady (time varying) component of the motions of the ship and the free surface elevation are small and of order $\xi$. In this case (3.7) can be broken down into $O(1)$ components that do not vary with time and depend on the mean position of the hull and free surface, $O(\xi)$ components which are linear in $\xi$, $O(\xi^2)$ components which are quadratic in $\xi$, and other higher order components. The added resistance is defined as the second order component of this total resistance. Any higher or lower order terms of equation (3.7) than quadratic are thus discarded for added resistance calculations.
Part II
Contributions

Simulation of an IACC yacht beating to windward
CHAPTER 4

VISCOUS EFFECTS

This chapter will present a method for incorporating viscous effects and their interaction with the potential flow in the presence of a free surface. A direct viscous-inviscid interaction algorithm is developed using the Rankine panel method and an integral turbulent boundary-layer method. Some numerical results are presented for an IACC sailing yacht.

4.1 Introduction

In 1872, William Froude made a bold assumption which has proved to be of the utmost practical importance, but of somewhat inadequate theoretical justification. Faced with the problem that the forces acting on a body moving through a viscous fluid are a function of both the Reynolds number and the Froude number, he speculated that the total resistance can be treated as the independent sum of these two components. This assumption is essential for model tank testing theory, since it is impossible to operate a ship and a model at the same Froude number and Reynolds number simultaneously [25, 38].

The Froude hypothesis continues to be relied upon, even in computational fluid dynamics, due the efficiency and robustness of inviscid flow calculation techniques such as panel methods. It has, therefore, become standard design practice to use
potential flow theory to analyze the flow, and then compensate for viscous effects \textit{a posteriori} using some empirical formula such as the ITTC friction line. The advantage of this approach is that complex problems may be solved quickly, without severely compromising accuracy. It does, however, ignore the interaction between viscous and wave effects, which can be quite important.

The solution of the Navier-Stokes equations requires considerable computational effort, especially in the presence of a free surface. With the present computer power, calculations for three-dimensional steady flows are just now becoming feasible, although the most promising methods use a composite flow description (zonal approach) rather than directly solving the problem of a viscous flow subject to free surface boundary conditions [22, 4, 6].

Viscous flows can be computed more efficiently using viscous-inviscid interaction methods. Such methods are less general than Navier-Stokes solvers, but have been used with success in the field of aerodynamics. They make use of Prandtl's observation that for high Reynolds number external flows, viscous effects are confined to a thin boundary-layer which forms along the the body. Hence, the viscous flow need not be solved in the entire fluid domain in order to capture the effects of viscosity.

This chapter will present a viscous-inviscid interaction method, which uses the Rankine panel method described in chapter 2 to solve the outer, inviscid flow, and an integral boundary-layer method to solve the inner, viscous flow. The solutions in the two regions are coupled through the boundary conditions of each problem. Because each part of the flow is represented by a simplified model, the overall computational cost is much less than solving the complete problem directly.

For the ship resistance problem, the coupling of the solutions in the two flow regions will provide more accurate values of wave-making resistance and wave patterns. Potential flow methods are known, for example, to commonly over-predict the wave elevation near the stern. In addition, the coupling will provide a good prediction of the frictional resistance of the ship. Naval Architects have traditionally assumed that the frictional resistance of a ship is equal to that of any body of equal wetted area. This, although practical, is obviously a crude approximation, since the presence of pressure
4.2. The Boundary-Layer Model

Gradients on a three-dimensional body strongly affect the boundary-layer. The incorporation of a numerical boundary-layer model, would therefore enable a computer code like SWAN to give a good prediction of the total resistance of a seagoing vessel.

The boundary-layer model which was used to account for the viscous effects is based on an integral turbulent boundary-layer method and is described in section 4.2. A description of the method used to modify the potential flow boundary conditions to account for the presence of the boundary layer is given in section 4.3, while section 4.4 presents the coupling algorithm which was used to capture the interaction between viscous and potential flows. Some results of this coupling are presented in section 4.6 and, finally, conclusions are drawn in section 4.7.

4.2 The Boundary-Layer Model

As will be shown in section 4.3, the Rankine panel method boundary conditions may be altered by specifying a new effective normal fluid velocity on the body and the linearized free surface. This effective normal velocity depends on only the displacement thickness $\delta^*$, and the fluid velocity of the potential flow just outside the boundary-layer. Integral boundary-layer methods are capable of providing accurate estimates of integral quantities such as displacement thickness, without having to solve for the velocities in the entire boundary-layer. Such a method is therefore an appropriate boundary-layer model to be coupled with the potential flow panel method.

A brief outline of the formulation of the integral turbulent boundary-layer equations is given below. For a more detailed discussion, however, the reader should refer to the report by Green, Weeks and Brooman [11].

4.2.1 Boundary-Layer Equations

The boundary-layer that forms along bodies at high Reynolds numbers, may be assumed thin compared to the characteristic length of the body. Outside this region, the fluid behaves very much in accordance with potential flow theory. Prandtl showed that the pressure can be assumed constant across the boundary-layer and that diffu-
sion can be neglected, except in the direction normal to the wall. The Navier-Stokes equations are then reduced to thin shear layer equations.

The boundary-layer is defined in terms of three independent integral parameters: momentum thickness $\theta$, shape parameter $H = \delta^* / \theta$, and entrainment coefficient $C_E$. The two-dimensional integral boundary-layer equations are derived by integrating the thin shear layer and continuity equations across the boundary-layer in the direction normal to the body. Hence obtained, are the momentum integral equation (4.1), the entrainment equation (4.2) and an equation for the streamwise rate of change of entrainment coefficient (4.3). This last equation explicitly represents the balance between the advection, production, diffusion and dissipation of turbulent kinetic energy.

$$\frac{d}{d\xi}(r\theta) = \frac{rC_f}{2} - (H + 2)\frac{r\theta dU_e}{U_e}$$

$$\frac{\theta dH}{d\xi} = dH \left[ C_E - H_1 \left\{ C_f \left( \frac{C_f}{2} - (H + 1)\frac{\theta dU_e}{U_e} \right) \right\} \right]$$

$$\frac{\theta dC_E}{d\xi} = F \left[ \frac{2.8}{H + H_1} \left( (C_f)^{\frac{1}{2}} - \lambda C_f^{\frac{1}{2}} \right) + \left( \frac{\theta dU_e}{U_e} \right)_{EQ} - \left( \frac{\theta dU_e}{U_e} \right) \right]$$

For a description of the quantities in the above equations, refer to the summary of the integral turbulent boundary-layer method given in appendix A. Closure relations necessary for the estimation of the secondary, dependent parameters are also given in the same appendix.

The development of the boundary-layer in a given pressure distribution is predicted by the integration of the above three simultaneous ordinary differential equations. The hyperbolic nature of the problem suggests a forward numerical integration with given boundary conditions upstream. The wake downstream of the trailing edge of the body is treated by continuing the integration of the same equations. The only changes to the method are to set the skin friction coefficients, $C_f$ and $C_{f_0}$, equal to zero and to double the dissipation length scale.

The validity of the above boundary-layer equations is limited to two-dimensional and axisymmetric flows. In order to be used for three-dimensional flows, it is necessary
to assume that the mean velocity profiles in the boundary-layer are collateral\(^1\). It is then sufficient to solve the two-dimensional integral boundary-layer equations along the streamlines of the inviscid flow.

Crossflow was neglected in the present analysis for simplicity. It would be possible to take the full three-dimensional integral boundary-layer equations into account, as has been done recently by Milewski [28] for the unbounded fluid case. This would add some complication to the problem but is feasible. It is, however, left as a future extension of this work. A two-dimensional formulation of the boundary-layer is expected to give a good first estimate of the viscous effects and their interaction with the wave flow past a three-dimensional ship.

### 4.2.2 Boundary Conditions

In order to close the problem, initial values of the main parameters must be assumed at a given transition point downstream of the leading edge. A flat plate 1/7-power law is used to obtain these values, which approximates the velocity distribution within the boundary-layer as follows

\[
\frac{u}{U_e} = \left(\frac{y}{\delta}\right)^{1/7}
\]  

(4.4)

By integrating the Karman relation, the dependence of the main integral parameters is, therefore, found to be

\[
\delta = 0.373 \xi R_\xi^{-1/5}
\]  

(4.5)

\[
\theta = 0.0363 \xi R_\xi^{-1/5}
\]  

(4.6)

\[
C_f = 0.0592 R_\xi^{-1/5}
\]  

(4.7)

\[
\delta^* = 0.0467 \xi R_\xi^{-1/5}
\]  

(4.8)

where \( R_\xi = \rho U_e / \mu_e \xi \) is the local Reynolds number.

---

\(^1\)Such flows, where there is no rotation of the velocity vector along a normal to the body surface, are said to have zero crossflow.
4.3 Inviscid Flow Compensation for Viscosity

In order to account for the presence of viscosity, the quantities determined by the boundary-layer method described in section 4.2 are used to modify the potential flow solution provided by the Rankine panel method.

The coupling between the two methods is enforced through the boundary conditions at the edge of the boundary-layer. The inviscid flow provides the pressure field that is imposed upon the boundary-layer, while the boundary-layer displaces the outer flow away from the body.

The problem is that the viscous model requires the velocity distribution evaluated at the initially unknown edge of the boundary layer, as input. This is solved by taking the velocity distribution at the exact body and free surface wake as a first approximation, determining the size of the boundary-layer and then appropriately compensating for viscosity to get another potential flow solution. This procedure is repeated until a converged solution is reached.

The present section will derive the modified body and free surface boundary conditions, used in the inviscid flow problem to account for the presence of the boundary-layer.

4.3.1 Breathing Velocity

Although it is possible to use the displacement thickness to directly alter the dimension of the body after each iteration, the variable body geometry required at each step is not practical for numerical calculations. Instead, the displacement thickness is used to compute an effective normal velocity on the body (or the wake), which produces the required effect on the flow outside the boundary-layer. Hence, the original body geometry but a different body boundary condition is used in the potential flow panel method. Similarly, the kinematic free surface boundary condition in the wake is modified to account for this effective normal velocity.

The effective normal velocity on the body, or “breathing velocity” as it is often termed, is taken as the transverse velocity just outside the boundary-layer, $V_e$. An
expression for $V_e$ may be obtained by integrating the continuity equation across the boundary-layer and using the definition of the displacement thickness.

$$V_e = -\int_0^\delta \frac{\partial u}{\partial \xi} d\eta$$

$$= U_e \frac{d\delta}{d\xi} - \frac{d}{d\xi} \int_0^\delta u d\eta$$

$$= U_e \frac{d\delta}{d\xi} - \frac{d}{d\xi} (U_e (\delta - \delta^*))$$

$$= \frac{d}{d\xi} (U_e \delta^*) - \delta \frac{dU_e}{d\xi} \quad (4.9)$$

where $\xi$ and $\eta$ are the local coordinates in the streamwise direction and normal to the body respectively, and $U_e$ is the velocity in the streamwise direction. The boundary-layer thickness $\delta$, is taken as the distance from the body where $u/U_e = 0.995$ and is easily determined from the mass-flow shape parameter, $H_1 = (\delta - \delta^*)/\theta$. (See Appendix A)

### 4.3.2 Body Boundary Condition

The body boundary condition in inviscid flow is considered to be the “no-flux” condition, and has been expressed by (2.5).

This boundary condition is modified to allow a flux equal to the breathing velocity, as follows

$$\frac{\partial \Psi}{\partial n} = (\tilde{W} + \frac{\partial \delta}{\partial t}) \cdot \hat{n} + V_e \quad (4.10)$$

There is a choice of incorporating the extra normal velocity $V_e$, in the body boundary condition of either the basis, or the perturbation flow. As observed by Kring [18], allowing a normal flux through the aft part of the hull in the basis flow presents a good linearization for ships with transom sterns. He found that the aft stagnation pressure in the double body flow for such vessels caused poor convergence, and used an “aspiration model” to remove the aft stagnation pressure from the basis flow. The breathing velocity, which is greater near the stern, would have the same effect as Kring’s method, thus attaining better numerical behavior.
In the present Rankine panel method, however, the double-body flow with zero flux through the boundaries has properties which are needed for the evaluation of the m-terms\(^2\). Modifying the basis flow is not, therefore, formally correct and will be avoided.

### 4.3.3 Kinematic Free Surface Boundary Condition

The kinematic free surface boundary condition has been given in (2.3) and states that a fluid particle on the free surface always remains on the free surface. This implies that there is no flux across the exact position of the free surface. In order to account for the presence of a boundary layer in the wake, the above condition is modified to allow a flux per unit area across the free surface, equal to the breathing velocity.

\[
\left[ \frac{\partial}{\partial t} + (\nabla \psi - \vec{W}) \cdot \nabla \right] [z - \zeta] = V_e
\]

Equation (4.11) is approximate and cannot be imposed at the exact instantaneous position of the free surface, \(\zeta(x, y, t)\). It is, however, valid when applied to the linear problem as given in section 2.1.5, where the boundary condition is linearized about a basis flow \(\Phi\), and applied on the \(z = 0\) plane. The modification to the linearized kinematic free surface boundary condition is then derived in terms of the breathing velocity as follows

\[
\frac{\partial \zeta}{\partial t} - (\vec{W} - \nabla \Phi) \cdot \nabla \zeta = \frac{\partial^2 \Phi}{\partial z^2} \zeta + \frac{\partial \varphi}{\partial z} - V_e
\]

The above equation replaces (2.8) as the perturbation flow kinematic boundary condition. The dynamic free surface boundary condition remains unaltered, since the pressure across the boundary-layer is assumed to be constant.

\(^2\)As shown by Nakos [30], the evaluation of second order derivatives of the basis flow potential on the hull may be avoided when calculating the influence of the m-terms, by making use of a theorem due to Ogilvie and Tuck [37]. This theorem may be used, however, only under the condition of zero flux of the basis flow through the surface of the body. The double-body flow does indeed satisfy this condition, but the aspiration model does not.
4.4 The Coupling Algorithm

The procedure used to combine the solutions of the flow in the viscous and inviscid regions is known as the coupling algorithm. Algorithms which have been used in two dimensions include direct, inverse, semi-inverse, quasi-simultaneous, and simultaneous coupling. Since the present method is a first attempt at the coupling of viscosity with inviscid three-dimensional free surface flows, the simplest approach, a direct coupling algorithm, was used. This method has been proven to converge, provided separation is not encountered. In the field of aeronautical engineering, full three-dimensional direct coupling methods have been developed by Lazareff and Le Balleur [23] for transonic flow over finite wings.

A schematic description of the method used in this study to combine the viscous and inviscid solutions is presented in figure 4-1. This coupling algorithm is described in more detail below.

- **Potential flow solution**: The potential flow panel method can produce a pressure distribution on the body by ignoring the effect of the boundary-layer,
which will be relatively close to the pressure distribution of the full viscous flow. The velocities on the body and free surface are then recorded for use by the boundary-layer model.

- **Streamline tracing**: The output velocities from the panel method are used to trace streamlines along the body and into the wake. The velocity distribution on each streamline is recorded for input to the boundary-layer model.

- **Viscous flow solution**: The integral turbulent boundary-layer method (appendix A) uses the velocity distribution on each streamline to produce a solution. This solution consists of a distribution of the main integral parameters along each streamline which will in turn be used to modify the potential flow solution.

- **Modification of the potential flow problem**: As shown in section 4.3, basic integral parameters of the boundary-layer solution may be used to compute an effective normal velocity on the body and the free surface. This normal velocity, to be imposed by the inviscid flow boundary conditions on each panel of the input geometry, is found by interpolating $V_e$ (4.9) between streamlines.

- **Iteration**: The potential flow is then solved once again, taking breathing velocities into account and new streamlines are traced. This procedure is repeated, typically two to three times, until a converged solution is reached.

### 4.5 Viscous Force Calculations

This section reviews two methods for calculating the viscous resistance of a three-dimensional body using the integral turbulent boundary-layer method described above. The first one is based on a shear stress integration and the other is based on conservation of momentum considerations.
4.5.1 Shear Stress Integration

One of the parameters of the integral turbulent boundary-layer method is the skin friction coefficient $C_f$, which is defined as the shear stress on the body surface normalized by $\frac{1}{2} \rho U^2$. The integration of this shear stress over the wetted surface of the body results in the total viscous force acting on the hull.

$$F_v = \frac{1}{2} \rho \int \int_{S_B} U_e^2 C_f \cos \alpha \ dS$$

(4.13)

where $\alpha$ is the angle between the x-axis and the tangential vector to the body in the streamwise direction.

4.5.2 Conservation of Momentum

Alternatively, the two-dimensional drag associated with a single streamline may be found by conservation of momentum and then integrated over the width of the viscous flow to get the total viscous drag.

What follows is the derivation of a method for calculating the two-dimensional viscous drag, given the integral parameters of the boundary-layer up to a point in the wake downstream of the body. The control volume considered is shown in figure 4-2 and extends longitudinally from far upstream of the body to a point in the wake. The thickness is constant and equal to the thickness of the boundary-layer $\delta$, at the downstream end of the control volume. Let $U_e$, $\delta$, $\delta^*$, and $\theta$ denote the quantities at the downstream end of the control volume. $U_\infty$ is the free stream velocity. The coordinates $\xi$, $\eta$, and the fluid velocities at the edge of the boundary layer $U$, $V$ are, respectively, locally tangent and locally normal to the body.

By applying the principles of conservation of mass and momentum in the control volume, and using the definition of the displacement thickness (A.1), equations (4.14) and (4.15) follow.

$$\int_{0}^{l} V \ d\xi = \int_{0}^{\delta} [U_\infty - u(l, \eta)] \ d\eta$$

$$= U_e \delta^* - (U_e - U_\infty) \delta$$

(4.14)
Chapter 4. Viscous Effects

Figure 4-2: Control volume used for the viscous drag calculation

\[ F_v = \int_0^\delta \rho \left( U_\infty^2 - u^2(l, \eta) \right) \, d\eta - \int_0^l \rho \, U(\xi) \, V(\xi) \, d\xi \]  \hspace{1cm} (4.15)

The final result for the viscous drag (4.16) is obtained by integrating the last term of (4.15) by parts and substituting the definition of the mass flux across the top of the control volume (4.14) and the definition of the momentum thickness (A.2).

\[ F = \rho U_c^2 \delta + \rho U_\infty (U_\infty - U_e) \delta + \rho \int_0^l \frac{dU}{d\xi} (U \delta^* - U \delta + U_\infty \delta) d\xi \]  \hspace{1cm} (4.16)

4.6 Results

The method of coupling the viscous and the inviscid flow regions has been implemented by Martinot-Lagarde [27] for some simple two-dimensional and axisymmetric geometries in an unbounded fluid where the potential flow can be found analytically. Converged results are thus published for a flat plate, a two-dimensional Karman-Trefftz strut and an ellipsoid of revolution.

In this section, results will be presented for a more complex geometry in the presence of a free surface, where it is necessary to solve the potential flow using a panel method. The actual hull which will be examined is that of an International America’s Cup Class sailing yacht. This type of hull was selected not only because the 1995 America’s Cup was a major motivation for this study, but also because such hulls are very streamlined and no separation is expected to occur. The occurrence of separation is known to cause convergence problems in direct coupling algorithms such as the present one.
4.6.1 Streamline Tracing

As explained in section 4.4, a panel method was used to determine the fluid velocities on the hull and free surface, which are necessary in order to trace streamlines. Figure 4-3 shows 10 streamlines traced on an IACC hull traveling through an ideal fluid at Froude number 0.347 (9 knots).

![Figure 4-3: Streamlines traced on an IACC hull (not to scale)](image)

For two streamlines at transverse extremes of the wake, the velocity distributions that were used as input to the integral boundary-layer method are shown in figure 4-4. The results for three iterations are presented.

4.6.2 Boundary-Layer Parameters

By tracing streamlines, recording the fluid velocity along them and then applying the two-dimensional viscous model for each one of them, the integral parameters are found on each streamline. Figures 4-5 through 4-8 show the displacement thickness, shape parameter, breathing velocity, and skin friction coefficient on two streamlines.

By observing these results it can be seen that after three iterations the interaction between potential and viscous flows has been fully captured. It is interesting to note that for the streamline near the centerline of the body, the boundary-layer parameters are accurate after the first iteration because the path of this streamline is not greatly affected by the presence of the boundary-layer. Near the waterline, however, there is more freedom for the streamlines to change due to the presence of the free surface. The effect of viscosity on the streamlines is thus greater.
Chapter 4. Viscous Effects

Figure 4-4: Velocity distribution on two streamlines

Figure 4-5: Displacement thickness on two streamlines
Figure 4-6: Shape parameter on two streamlines

Figure 4-7: Breathing velocity on two streamlines
4.6.3 Form Factor Calculations

It is common practice among Naval Architects to calculate the frictional resistance coefficient of a vessel by assuming that it is independent of Froude number and that it is given by some constant multiple \((1 + k)\), of a known friction line such as the 1957 ITTC. The factor \(k\) accounts for the three-dimensional form, and is appropriately termed the form factor.

This very practical hypothesis, due to Hughes, is universally applied when extrapolating experimental data from model to full scale. In reality, however, due to the interaction between the boundary-layer and the wave flow, the viscous component of resistance is not expected to be a constant multiple of the frictional resistance of a flat plate. It is possible to investigate this interaction by using the present method to examine the viscous force sensitivity to Froude number.

Using the boundary-layer model described in this chapter, an estimate of the viscous resistance coefficient \(C_v\) of the hull is obtained (see section 4.5). The form factor is then given by

\[
k = \frac{C_v}{C_{ITTC}} - 1
\] (4.17)
where $C_{ITTC}$ is the 1957 ITTC friction line at the given Reynolds number.

Figure 4-9 shows the form factor thus calculated for a range of Froude numbers. The disagreement with Hughes' constant form factor hypothesis is quite obvious. The negative form factor for low Froude numbers might seem counter-intuitive but it is consistent with what has been experimentally observed for many streamlined vessels such as IACC yachts. The reason is that the 1957 ITTC friction line does not represent the frictional resistance of a flat plate, but rather is a model-ship correlation line and includes the form drag of an average ship hull.

All viscous force calculations in this section were performed by shear stress integration (4.13) because it encountered less numerical difficulties than the conservation of momentum method. The difficulties of the latter method were due to the fact that it was numerically difficult to precisely define the lateral extent of the viscous wake and to evaluate the momentum flux across the boundary-layer in this region. Since there would be no need to trace streamlines, this problem would have been avoided if the full three-dimensional integral boundary-layer equations were coupled with the panel method. In any case, the direct stress integration method has been proved sufficient to provide satisfactory results.
4.6.4 Wave Patterns and Forces

The influence of the viscous boundary-layer on the inviscid flow for the hull of the preceding section is assessed below. Solutions are produced both with and without viscous effects included. They are then compared to determine the extent by which the wave patterns and wave resistance are affected from the presence of the boundary layer.

The effect of viscosity on the wave pattern may be observed in figure 4-10(a) for Froude number 0.347. As expected, the strongest effect is in the area immediately downstream of the stern. More specifically, the wave elevation near the stern is reduced and as a result the pressure at the aft part of the hull is also lower. Therefore, the net effect on the wave forces is an increase in drag.

Over a range of Froude numbers a similar effect is observed, as shown in figure 4-10(b). The wave resistance in general increases, more notably in the lower to middle Froude number range. This difference in resistance could be of considerable importance for some applications, such as the design of America’s Cup yachts where differences of the order of a few percent are significant.
4.7 Conclusions and Recommendations

A method of coupling an integral turbulent boundary-layer method with a Rankine panel method has been devised and implemented. The benefits of this approach is that both major components of the total drag of a ship, the viscous and wave drag, may be estimated with better accuracy than if they were considered separately.

The viscous force calculations demonstrate shortcomings in the traditional approach of assuming that a ship's viscous resistance coefficient is equal to a constant multiple of the resistance coefficient of a flat plate. In addition, the wave force calculations suggest that the interaction between the free surface and the viscous boundary-layer can be important at certain Froude numbers.

The method is, however, computationally intensive, as it is necessary to solve the inviscid flow problem at least two times in order to converge to a solution that satisfies both the integral turbulent boundary-layer equations and the potential flow boundary value problem.

Also, there are several restrictions of this method, which invite the attention of future work.

- Problems are expected to occur if separation is encountered in the flow. The direct coupling algorithm used is known to fail in such situations. The solution would be to employ a simultaneous coupling algorithm, as in the work of Milewski [28] for an unbounded fluid. But even then, the flow past bluff bodies with open separation would not be able to be treated.

- Currently, the method assumes negligible crossflow, which can be a rather severe approximation for realistic flows past ship hulls. The three-dimensional integral boundary-layer equations could be used to rectify this situation, at the expense of a more elaborate boundary-layer solution scheme.

- For all the above analysis, a steady flow is assumed. If the viscous effects are to be included for unsteady flows, the integral boundary-layer equations need to be solved at each time step of the panel method. Unless a more intelligent
coupling scheme is devised, this would be too computationally demanding for real applications.
CHAPTER 5

LIFTING SURFACES

This chapter presents an extension of the Rankine panel method for free surface flows with lift.

5.1 Introduction

Some special types of ships, such as sailboats and hydrofoil craft, operate with their hulls designed to produce a significant amount of lift. In addition, multi-hulled vessels, such as catamarans, may have a certain amount of interaction between their hulls, which cannot be accurately predicted without considering them as lifting surfaces. The ability to include lifting surfaces is therefore essential, and it needs to be included in any method intended for the evaluation of such complex hull-forms.

Panel methods were first applied to the aeronautical industry, and it was, therefore, not long before they were further developed to take the lift and induced drag of airplane wings into account [12]. Such methods have been adopted by Naval Architects in the past, and have been directly applied to the flow past ship hulls, neglecting the presence of the free surface\(^1\). This approach was used by Greeley and Cross-Whiter [10] to design keels for the 1987 America’s Cup campaign, for example.

\(^1\) A zero Froude number approximation is needed for this approach, which replaces the free surface by a rigid wall.
More recently, several steady flow panel methods have been extended to include the interaction of lift-producing hulls with the free surface [49, 41, 14]. All these methods introduce a lift force by distributing vortices or dipoles on the body or the mean camber line. In addition, they impose a Kutta condition of flow tangency or pressure equality at the trailing edge, and introduce a trailing vortex system shed into the flow to satisfy Kelvin’s theorem.

This chapter will use a similar approach to solve the steady or unsteady lifting problem in the time-domain. Section 5.2 explains how to incorporate lifting surfaces in the formulation of the boundary value problem. The numerical implementation is given in section 5.3, and some simple cases are examined for validation purposes in section 5.4.

5.2 Formulation

Section 2.1 has formulated the problem of the flow without circulation. Extending the numerical implementation described therein, changes are needed in both the basis and perturbation flows.

5.2.1 Wake Condition

The wake behind the lifting surface is modeled by a free vortex sheet. This sheet is considered infinitesimally thin and is composed of two surfaces, $S_W^+$ and $S_W^-$, which occupy the same position but have opposite normal direction.

The wake surface is assumed to be fixed in the vessel coordinate system and its shape follows by tracing the lifting surface trailing edge directly downstream. The two surfaces representing the wake are combined into a single jump surface, $S_W$, which in the present method is represented by a dipole sheet. In what follows, $\Delta$ is the operator which denotes the jump in a quantity across the wake, and the superscripts “+” and “−” denote quantities on the surfaces $S_W^+$ and $S_W^-$, respectively.

Relating circulation around the body to the potential jump across the wake, Morino [29] proposed a linear Kutta condition which specifies the strength of the
dipole sheet in the wake. This condition states that the potential jump in the wake just downstream of the trailing edge must equal the difference in potential on the body on each side of the trailing edge. This is equivalent to a statement of continuity of potential from the body into the wake. A similar idea is successfully used for the treatment of deep transom sterns in chapter 7.

If the trailing edge has finite thickness, the collocation points of the body panels on opposite sides of the trailing edge may have different values of free-stream potential. It is then necessary to apply a correction to the Morino condition, as proposed by Lee [24], which requires the potential jump in the wake to be equal to the difference in total potential at the collocation points of the panels at the trailing edge.

\[ \Delta \Psi(\vec{x}_W, t) = \Psi(\vec{x}_B, t)^+ - \Psi(\vec{x}_B, t)^- - \vec{W} \cdot \vec{r}_{TE} \]  \hspace{1cm} (5.1)

where \( \vec{x}_W \in S_W \), and \( \vec{x}_B \in S_B \) at the intersection of \( S_B \) with \( S_W \). \( \vec{r}_{TE} \) is the vector joining the collocation points of the two trailing edge panels.

The wake can sustain no forces so there must be no jump in pressure, \( p(\vec{x}, t) \), across the sheet

\[ \Delta p = p(\vec{x}, t)^+ - p(\vec{x}, t)^- = 0 \]  \hspace{1cm} (5.2)

Using Bernoulli’s equation and linearizing about the free stream, an expression for the potential jump \( \Delta \Psi(\vec{x}, t) \) in the wake, may be obtained.

\[ \frac{\partial \Delta \Psi}{\partial t} - \vec{W} \cdot \nabla(\Delta \Psi) = 0 \]  \hspace{1cm} (5.3)

Although accurate for two-dimensional sections with small exit angles, this linearization may not be satisfactory for three dimensional problems with significant cross-flow [24]. In this case, a non-linear Kutta condition is required, explicitly stating that the pressure jump in the wake must vanish.

However, for high aspect ratio lifting surfaces such as sailboat keels and rudders, the cross-flow is not expected to have a significant effect at most sections of the foil. Indeed, examination of typical foils at an angle of attack of less than 10 degrees
revealed that the linear Morino condition also forced the pressure jump at the trailing edge very close to zero in most cases.

One exception is the vicinity of the intersection of the vortex wake with the free surface, where there is an inconsistency between the wake and free surface condition linearizations. The linear Morino condition was derived from the linearization of the flow about the free stream, with the intention of setting the pressure jump in the wake equal to zero. But in general, the basis flow which is used to linearize the free surface conditions leads to a finite pressure jump across the wake. Through the dynamic free surface condition, this translates to a jump in the wave elevation. It is interesting to note is that this jump is also experimentally observed in real flows.

For lower aspect ratio foils which are not expected to operate under high loading, such as the demi-hulls of a catamaran, it has been shown by Kring [19] that the above linearization is satisfactory.

5.2.2 Basis Flow

As seen in section 2.1.4, the source formulation for the solution of the double-body flow may not be used when circulation is present. Instead, the potential formulation (2.7) is modified to account for the presence of a wake sheet.

Collapsing the wake surfaces \( S_W^+ \) and \( S_W^- \) into a single surface, and imposing a continuous normal velocity across it, Green’s second identity yields an expression for the unknown potential on the body;

\[
2\pi \Phi(\vec{x}) - \int_{S_B \cup S_B^*} (\vec{V} \cdot \hat{n}) \ G(\vec{x}'; \vec{x}) \ dx' + \\
\int_{S_B \cup S_B^*} \Phi(\vec{x}') \ \frac{\partial G(\vec{x}'; \vec{x})}{\partial n} \ dx' + \\
\int_{S_W} \Delta \Phi(\vec{x}') \ \frac{\partial G(\vec{x}'; \vec{x})}{\partial n^+} \ dx' = 0 \quad (5.4)
\]

The potential jump on the dipole sheet is also unknown, but is determined in terms of the potential on the body through the Morino condition (5.1). Since the flow is steady, the jump is merely required to be constant in the streamwise direction
5.3. Numerical Implementation

according to (5.3).

5.2.3 Wave Flow

The linearization of the problem about the basis flow does not differ from the case without lift, with the body and free surface boundary conditions as given in section 2.1.5.

In a similar manner as for the basis blow, the governing equations of the wave flow expressed in boundary-integral form (2.12) may be modified to account for the presence of a vortex wake sheet;

\[ 2\pi \varphi (\vec{x}) = \int \int_{S_p \cup S_B} \frac{\partial \varphi (\vec{x}')}{\partial n} G(\vec{x}', \vec{x}) \, dx' + \int \int_{S_p \cup S_B} \varphi (\vec{x}') \frac{\partial G(\vec{x}', \vec{x})}{\partial n} \, dx' + \int \int_{S_w} \Delta \varphi (\vec{x}') \frac{\partial G(\vec{x}', \vec{x})}{\partial n^+} \, dx' = 0 \]  

where \( \vec{x} \in (S_p \cup S_B) \).

The jump in potential in the wake is time-dependent in this case, as dictated by (5.3).

5.3 Numerical Implementation

The numerical implementation of the method including vorticity is along the lines given in section 2.2. There is an additional number of unknowns in the integral equation, equal to the number of panels immediately aft of the trailing edge of the lifting surface. Each of those panels is associated with a pair of panels on the body, and the additional equations required to find a solution follow from the Morino condition (5.1).

The potential jump on the rest of the panels of the wake is determined separately from an explicit Euler integration of (5.3), with upwind differencing for the evaluation of the spatial derivatives. The upwinding adds some damping to the vortex wake
system, but this does not affect the solution in any visible way over the range of practical wave frequencies tested. The free surface, which is much more critical to the solution, is still free of damping. Another alternative, which is to use central differencing for the evaluation of the spatial derivatives in the wake, is impractical due to the severe restrictions in panel length and time step in order to achieve stability.

The actual value of the potential on the two sides of the wake need not be determined for this problem, although it would be possible, as described in section 6.1.

In some cases of problems with lift that were studied, the bi-quadratic spline functions on the body were found to have an oscillatory behavior about the true solution. This was observed especially for foils with sharp leading edges under significant angles of attack and was due to the inability of the splines to capture the rapidly varying flow at that point. The solution for these cases was to use panels of constant strength, with cosine spacing at the leading edge.

5.4 Validation

Some results are presented in this section in order to validate the implementation of lift in the method. First, a foil with an elliptical planform and Karman-Trefftz sections was tested in an infinite fluid and compared to an analytical solution. Then, a surface piercing foil was examined and the results were compared to experimental observations.

5.4.1 Foil in Infinite Flow

A Karman-Trefftz section is obtained by a transformation in the complex plane of a circle $z$, of center $z_c$, which passes through a specified value, $a$, on the real axis.

$$
\zeta = \frac{\lambda a [(z + a)^\lambda + (z - a)^\lambda]}{(z + a)^\lambda - (z - a)^\lambda} \quad (5.6)
$$

$\lambda$ is a parameter.
The analytic solution of the potential flow past the section obtained by the transformation of (5.6) is known. Taking \( a = 1 \), the parameters \( z_c = x_c + iy_c \) and \( \lambda \) may be varied to produce a great variety of realistic-looking foil sections.

For a sufficiently high aspect ratio, the solution for a two-dimensional section may be extended to three dimensions by using Prandtl’s lifting line theory. The inflow at each section is therefore modified by an amount necessary to compensate for the downwash velocity. The analytical solution of the flow at each Karman-Trefftz section is obtained for a modified local angle of attack, which differs from the two-dimensional angle by \( \delta \alpha \);

\[
\delta \alpha = -\frac{C_{L(2D)}}{\pi (A + 2)}
\]  
(5.7)

where \( A \) is the aspect ratio, and \( C_{L(2D)} \) is the two-dimensional sectional lift coefficient.

The foil used in the present tests had a Karman-Trefftz section with parameters \( x_c = 0.1, y_c = 0.1, \tau = \pi (2 - \lambda) = 10 \), and an elliptical planform with an aspect ratio of \( A = 10.19 \). The angle of attack was 2 degrees.

The forces on the foil were computed by pressure integration over the body surface in the usual way. In addition, for this infinite fluid steady flow problem, the forces were also calculated by a Trefftz plane integration. The formulae for lift and drag calculation by this method are given below for convenience, and the details of their derivation may be found in a standard hydrodynamics textbook [34].

\[
D = \frac{1}{2} \rho \int_{s/2}^{s/2} \Delta \Phi \frac{\partial \Phi}{\partial z} \, dy
\]  
(5.8)

\[
L = \rho \, U \int_{s/2}^{s/2} \Delta \Phi \, dy
\]  
(5.9)

Figure 5-1 shows the actual shape of the Karman-Trefftz section used, and compares the pressure distribution at the mid-section as obtained numerically, with the analytic solution. It is evident that in this case the Rankine panel method captures the details of the flow very well.

The agreement, however, of the integrated forces on the foil (figure 5-2) are not as good as that of the pressure distribution at mid-chord. This, of course, is due to the
fact that lifting line theory ignores any spanwise velocities and therefore the three-dimensional method produces a different, presumably more correct, solution near the tips. Prandtl's theory also assumes an elliptic distribution of circulation, which is not exactly true for an elliptic planform. In fact, the geometry of the discretized foil which was used was not precisely elliptic, due to wake paneling difficulties at the tip.

Considering all of the above, the results are promising and in addition demonstrate that the solution is convergent and well behaved, even near the tip of the foil where problems might be expected to arise in a potential flow numerical method. This can also be seen in figure 5-3, which shows the spanwise distribution of the bound vorticity, and the potential along several chordwise strips of panels on the body. There is no numerical anomaly whatsoever near the tip.

A linearized, somewhat arbitrary wake geometry has been assumed in the above investigation. This only approximates the position of the wake in the real flow. More exact results could, in principle, be produced by tracing the wake at each time step according to the fluid velocity induced at each point. Work has been done in this
5.4. Validation

Figure 5-2: The convergence of the lift (a) and drag (b) coefficients for an elliptical foil with aspect ratio $A = 10.19$ and the section of figure 5-1, as computed by pressure and Trefftz plane integration. The analytic estimation is made by correcting the two-dimensional lift and drag coefficients along the foil using Prandtl’s lifting line theory.

Figure 5-3: Demonstration of good numerical behavior at the tip of an elliptical foil with $A = 10.19$. Shown, are the spanwise distribution of bound vorticity (a), and the chordwise distribution of potential at several sections of the foil (b). The section used is as described in figure 5-1.
5.4.2 Surface Piercing Foil

The next step towards validating the lifting model, was to run the code for a surface piercing hydrofoil and compare the results to prior experimental and numerical work.

The foil used for these tests had a rectangular planform with a span of 57 inches and a chord of $16\frac{3}{4}$ inches. The section shape was symmetric, with a thickness-to-
chord ratio of $T/C = 0.09$. This particular foil was chosen due to the availability of the *PACT*’95 syndicate’s experimental data from their America’s Cup testing program. Runs were performed at a yaw angle of $\alpha = 2^\circ$, over a range of speeds.

Observation of real flows just behind the trailing edge reveals that after a critical Froude number, a sharp jump occurs in the free surface elevation across the wake [3], [26]. This jump cannot be predicted by linear potential flow theory if both the wake and free surface conditions are linearized about the free stream. In this case the requirements of constant pressure across the wake and on the free surface would lead to a zero jump in the free surface across the wake.

In the present method, however, the free surface conditions are linearized about a basis flow, which also includes a jump in potential over the wake, $\Delta \Phi$. The theoretical elevation jump is therefore non-zero and given by

$$\Delta \zeta = \zeta^+ - \zeta^- = \frac{1}{2} \Delta (\nabla \Phi \cdot \nabla \Phi) + \Delta (\nabla \Phi \cdot \nabla \varphi)$$  \hspace{1cm} (5.10)

In addition, the free surface is divided into two separate spline sheets, which have a common boundary at the intersection with the wake sheet. There is no continuity condition across these free surface sheets, and hence the wave elevation is free to have a jump at this boundary without violating the assumptions of linear theory.

Figure 5-5 shows the wave pattern of the flow past the foil at Froude numbers of 0.3 and 1.0. The latter speed is beyond the critical Froude number and a jump in the free surface elevation at the trailing edge may be observed.

The lift coefficient and the free surface jump as a function of Froude number are shown in figure 5-6 and it can be seen that for this configuration, the critical Froude number where the free surface elevation jump occurs is approximately 0.4. A sharp increase in the lift coefficient occurs at this speed, followed by an apparent drop to a high Froude number limiting value. Experiments were available for the higher speeds shown, which agree well with the predictions of the numerical method.

The wave elevation near the trailing edge of the foil is further examined in figure 5-7, for Froude numbers ranging from 0.3 to 1.0. It can be seen that at the outset of
Chapter 5. Lifting Surfaces

Figure 5-5: Wave patterns of a surface piercing foil at $F_n = 0.3$ and $F_n = 1.0$

Figure 5-6: The lift coefficient (a) and the jump in wave elevation at the trailing edge (b) as a function of Froude number, for a surface piercing foil.
the flow regime where the elevation jump occurs, the suction side of the free surface at the tailing edge is at a higher position. As the speed increases, this area moves downstream and an area where the elevation is higher at the pressure side replaces it at the trailing edge. It is not documented whether in experimental observations the elevation jump continues for such a large distance downstream, but in real flows viscosity is certain to have a smoothing effect.
5.5 Conclusions

A linear three-dimensional time-domain Rankine panel method for the simultaneous prediction of free surface waves, lift and induced drag has been developed. Both steady and unsteady flows may be predicted, over a wide range of Froude numbers.

A linearized wake geometry was used, but the sensitivity of its actual position to the lift and induced drag of the body was quite low.

The numerical method displays good convergence properties with an increasing number of panels, and the agreement of the integrated forces with experiments for a surface piercing foil was satisfactory. In addition, the wave pattern was accurately resolved and shows a jump in the wave elevation at the trailing edge, as is observed for real flows.

The method is therefore considered mature for application to real problems such as sailing yachts and catamarans. Chapters 9 and 10 present results for such cases.
CHAPTER 6

THIN BODIES

The extension of the present Rankine Panel method to bodies with infinitesimally small thickness was motivated by the fact that there are numerous marine applications, such as the sails on a yacht, the damping plates on an offshore platform, and other appendages like rudders and winglets, which have a thickness very small compared to the overall dimensions of the structure.

To discretize such bodies on both sides and use the existing formulation would not only be inefficient from a computational efficiency point of view, but would also encounter fundamental numerical difficulties due to the proximity of the Rankine panel collocation points on the two surfaces of the thin body.

The solution is to treat the thin body as a single dipole sheet and re-formulate the boundary value problem to obtain an integral equation for the unknown strength of the dipole distribution. The problem is formulated in section 6.1 and the numerical implementation is given in section 6.2. The method is validated with some examples in section 6.3.

6.1 Formulation

Sections 2.1 and 5.2 have formulated the problem for bodies of finite thickness. An extension to thin bodies follows.
Chapter 6. Thin Bodies

The body is considered infinitesimally thin and is composed of two surfaces, $S^+_P$ and $S^-_P$, which occupy the same position but have opposite normal direction. As with the case of the vortex wake (section 5.2), these two surfaces are combined into a single jump surface $S_P$, which in the present method is represented by a dipole sheet.

Extending the notation of chapter 5, $\Delta$ is the operator which denotes the jump in a quantity across the thin body, and the superscripts "+" and "−" denote quantities on the surfaces $S^+_P$ and $S^-_P$, respectively.

If the thin body produces lift, the trailing vortex wake $S_W$, is treated exactly as described in chapter 5.

The usual boundary conditions, as described in section 2.1.3, apply in the presence of a free surface $S_F$, or a "thick" body $S_B$. For a thin body, the body boundary conditions become

$$\frac{\partial \Psi^-}{\partial n^+} = \frac{\partial \Psi^+}{\partial n^+} = \vec{W} \cdot \vec{n} \tag{6.1}$$

By applying Green’s second identity, collapsing the dipole sheets into single surfaces, and making use of the boundary conditions, the following integral equation is obtained;

$$- \int \int_{S_P \cup S_B} \frac{\partial \Psi(x')}{\partial n} G(x'; \vec{x}) \, dx' + \int \int_{S_P \cup S_B} \Psi(x') \frac{\partial G(x'; \vec{x})}{\partial n} \, dx' + \int \int_{S_P \cup S_W} \Delta \Psi(x') \frac{\partial G(x'; \vec{x})}{\partial n^+} \, dx' = \begin{cases} -2\pi(\Psi(\vec{x})^+ + \Psi(\vec{x})^-) & \vec{x} \epsilon (S_P \cup S_W) \\ -2\pi \Psi(\vec{x}) & \vec{x} \notin (S_P \cup S_W) \end{cases} \tag{6.2}$$

The RHS of (6.2) is different when the point $\vec{x}$ is on a dipole sheet because there is fluid on both sides of the singularity. In this case, the above equation cannot be used to determine the potential jump on $S_P$ because two extra unknowns, $\Psi^+$ and $\Psi^-$, have been introduced. Instead, the normal derivative of equation (6.2) is taken for points on $S_P$, and the body boundary condition (6.1) is used to eliminate the normal derivatives of the potential. A new boundary integral equation is hence obtained, which may be coupled with (6.2) for points on $S_B$ or $S_P$ in order to solve...
the complete problem;

\[
- \int \int_{S_F \cup S_B} \frac{\partial \Psi(x')}{\partial n} \frac{\partial G(x'; \bar{x})}{\partial p} \, dx' + \\
\int \int_{S_F \cup S_B} \Psi(x') \frac{\partial^2 G(x'; \bar{x})}{\partial p \partial n} \, dx' + \\
\int \int_{S_F \cup S_W} \Delta \Psi(x') \frac{\partial^2 G(x'; \bar{x})}{\partial p \partial n^+} \, dx' = -2\pi \left( \frac{\partial \Psi(x)^+}{\partial p} - \frac{\partial \Psi(x)^-}{\partial p} \right) \\
= -4\pi (\bar{W} + \frac{\partial \bar{z}}{\partial t}) \cdot \bar{p}
\]

(6.3)

where \( \bar{p} \) is the unit normal to \( S_F^+ \) at point \( \bar{x} \).

After the boundary value problem has been solved, the actual value of the potential on the thin body may be found by using equation (6.2), combined with the definition of the potential jump. This is necessary if the fluid velocities on the two sides of the thin body are to be determined.

### 6.2 Numerical Implementation

The base of the numerical algorithm is identical to that for a thick body, presented in sections 2.2 and 5.3.

After discretization, (6.3) gives one equation per panel on the thin surface, to determine the unknown jump in potential. In addition, one equation per panel for the other surfaces is produced by (6.2). Double derivatives of the Rankine source potential are required in order to obtain the coefficients of the new integral equation (6.3). These are automatically computed by the same algorithms [35] that are used for the evaluation of the coefficients of the original integral equation (6.2).

For lifting surfaces, constant panels with cosine spacing at the leading edge were needed for a non-oscillatory solution, similar to what was found for thick bodies in section 5.

It must be mentioned that the available subroutines for the evaluation of the influence coefficients do not calculate the velocity induced at a field point due to a distribution of normal dipoles of quadratic strength over a planar quadrilateral panel.
Therefore, the second term of the LHS of equation (6.3) cannot be readily evaluated on $S_F$ using the existing software. This can be easily addressed in the future, but for the purpose of this work the above term was evaluated assuming an equivalent distribution of constant strength on each panel of the bi-quadratic spline sheet on the body or the free surface. This does not cause significant errors unless the panels of the dipole sheet are too close to the bi-quadratic spline sheet, as shown in section 6.3.

6.3 Validation

Some results are presented in this section, both in an infinite fluid and in the presence of a free surface, in order to validate the implementation of the method.

6.3.1 Foil in Unbounded Fluid

An obvious first check to validate the method for thin bodies, is to determine how close the solution for a thin plate approaches to the limit of a foil of finite thickness of the same planform.

This test was indeed performed for a cambered foil of aspect ratio $A = 0.5$, and the results are shown in figure 6-1. The lift coefficient as a function of the thickness-to-chordlength ratio is shown along with the distribution of the potential jump over the chordlength of the plate at mid-span.

As expected, the numerical method has encounters difficulties when the two surfaces of the body are too close to each other. In this case, the proximity of the distribution of singularities on the two surfaces causes the influence coefficients in the boundary integral equation to be of very large magnitude. The numerical scheme tries to find a solution by subtracting large quantities, and after a point which depends on the accuracy of the particular machine on which the code is running, it fails.

But when the thin body is treated as a dipole sheet, no more such numerical problems are encountered and it can be seen that the value obtained in this case is clearly the low thickness-to-chordlength ratio limit of the “thick-body” method, before it fails.
6.3. Validation

Figure 6-1: The lift coefficient of a rectangular planform foil with $A = 0.5$ as a function of thickness-to-chordlength ratio (a), and the distribution of the potential jump over the chordlength of the plate at mid-span (b).

### 6.3.2 Submerged Horizontal Plate in Heave

A second test to validate the method for the unsteady problem was to analyze the flow associated with a heaving horizontal plate submerged under the free surface. The plate taken for this study was circular with diameter $d$, and was submerged at a depth equal to one radius under the $z = 0$ plane.

The heave added mass of a body in an unbounded fluid is given by

$$m_{33} = \rho \int \int_{S_B} \varphi \frac{\partial \varphi}{\partial n} dS$$

(6.4)

where $\varphi$ is the radiation potential of the body in heave [34].

For a thin plate where the surface $S_B$ is collapsed into a jump surface $S_P$ (see section 6.1), and for a unit translation velocity, the above condition becomes

$$m_{33} = \rho \int \int_{S_P} \Delta \varphi dS$$

(6.5)

The added mass may be, therefore, easily computed numerically using the present Rankine panel method to solve the infinite fluid problem for the plate translating.
Figure 6-2: Heaving circular plate. (a) The convergence of the heave added mass in an unbounded fluid and comparison with theory. (b) The wave elevation above the plate heaving with amplitude $A$, one radius ($d/2$) below the free surface. The frequency of heave is $\omega = 1.5(g/d)^{1/2}$.

with a steady unit velocity, and then directly evaluating (6.5).

The analytical solution for the above problem has been given by Lamb [21];

$$m_{33} = \rho \frac{1}{3} d^3$$ \hspace{1cm} (6.6)

The convergence of the method for an increasing number of panels on the circular disk is presented in figure 6-2(a) along with the theoretical value (6.6). The results are clearly convergent, and agree well with theory.

Adding a free surface to the problem, the wave pattern for the plate heaving at a normalized angular frequency $(\omega \sqrt{d/g})$ of 1.5 radians is shown in figure 6-2(b).

The added mass and damping of the plate heaving at a depth of one radius below the free surface may be calculated from the radiation potential by direct linear pressure integration. But the damping may also be found from the excitation force in the diffraction problem. More specifically, the Haskind relations [34] may be used to show that the heave damping coefficient of a body with a vertical axis of symmetry
Figure 6-3: The heave added mass (a) and damping (b) of a horizontal circular plate submerged at a depth of one radius. The damping is calculated from both the radiation and diffraction problems.

is equal to

$$b_{33} = \frac{\omega^3}{2\rho g^3} |X_3|^2$$

(6.7)

where $|X_3|$ is the vertical excitation force in regular waves of unit amplitude.

The added mass of the circular disk is presented in figure 6-3(a) as a function of frequency. The heave damping of the plate as found by both the above methods is in good agreement, as shown in figure 6-3(b), suggesting that the thin-body method gives plausible results for both the radiation and diffraction problems. The slight disagreement at low frequencies is due to the fact that, in order to obtain a domain size large enough to accommodate the wavelengths involved, a coarser grid with respect to the dimensions of the plate was used on the free surface in this case. As explained in section 6.2, the method as currently implemented can introduce errors when the dipole sheet is at a depth smaller than the order of magnitude of the dimensions of the free surface panels.

6.4 Conclusions

A linear three-dimensional time-domain Rankine panel method has been extended to include bodies of infinitesimally small thickness. Both steady and unsteady flows
may be predicted in the presence of a free surface or in an unbounded fluid.

The numerical method displays good convergence properties with an increasing number of panels, and the solution has been demonstrated to approach the limit of the method for bodies with decreasing but finite thickness.

Real problems, such as the flow past the sails of a sailboat, may now be tackled. Chapter 9 presents results for such a case.

Care must be taken when thin bodies are close to surfaces which are described in terms of bi-quadratic spline sheets, as the method currently approximates the influence of a quadratic distribution of normal dipoles on panels of such sheets with an equivalent constant dipole distribution.
CHAPTER 7

DEEP TRANSOM STERNs

A vessel with a deep transom stern is defined as one with a truncated afterbody, ending in a flat vertical section below the still waterline (figure 7-1). Since the great majority of modern ships have a stern of this type, it is imperative to be able to accurately model the flow in this region.

This chapter presents a method of resolving the numerical difficulties associated with the flow past a transom stern. The problem is formulated in section 7.1 and the numerical implementation is discussed in section 7.2. Some results are presented in section 7.3 for validation, while conclusions are drawn in section 7.4.

7.1 Formulation

For ships with transom sterns, the formulation of the problem as presented in section 2.1 is not complete. Since the flow separation at the sharp lower edge of the transom is triggered by viscous effects, the correct behavior must be explicitly enforced in the present potential flow mathematical model.

The real flow past a transom stern can exist in any one of three different regimes. The first regime occurs at low speeds or deeply immersed transoms, and is characterized by a stagnation pressure at the stern. In this case the transom remains wet and the viscous effects are dominant in the region.
The second regime occurs as speed is increased or the transom immersion is reduced. The flow detaches at the sharp transom edge and the transom itself remains dry. According to a rule of thumb \[42\], transition from the first regime to this type of flow often occurs at transom Froude numbers

\[ F_{nT} = \frac{U}{\sqrt{g z_T}} \approx 4.0 \]  

(7.1)

where \(z_T\) is the depth of immersion of the transom at zero speed. Most high-speed ships operate in this regime.

Finally, as transom immersion is further reduced so that the lower edge lies above the undisturbed free surface, the flow detaches at some point at the bottom of the hull before reaching the stern. Sailing yachts, for example, are very often designed to operate in this regime.

The first regime cannot be accurately modeled using a potential flow method and is, therefore, outside the scope of this study. In any event, such a flow is very inefficient, needs to be avoided by the ship designer, and thus rarely needs to be analyzed.

The third regime is highly non-linear, and cannot be predicted by the present linear method. A linear method can hope to achieve some approximate modeling of the flow by assuming, for example, that it detaches at the intersection of the body with the linearized free surface and by treating the flow as if it were in the second regime.

The present work will concentrate on transom flows in the second regime, where
the line of detachment is well defined, as shown in figure 7-1.

The free surface just aft of the transom must obey the dynamic and kinematic free surface conditions, and the mixed boundary-integral equation, just as at any other point on the free surface. But in order to enforce a flow detachment it may be necessary to impose some additional continuity conditions at that point.

- The wave elevation $\zeta_T$, should be equal to the instantaneous transom depth $z_T$.

$$\zeta_T = z_T$$ (7.2)

- Due to the smoothness of the solution, the wave slope at the transom may be imposed to be equal to the slope of the hull. The lack of this condition would allow a corner flow at the transom edge.

$$\frac{\partial \zeta_T}{\partial x} = \frac{\partial z_T}{\partial x}$$ (7.3)

$$\frac{\partial \zeta_T}{\partial y} = \frac{\partial z_T}{\partial y}$$ (7.4)

- At the transom, the velocity potential $\Psi_T$, and/or its normal derivative $\frac{\partial \Psi_T}{\partial n}$, should be equal on the body and in the free surface wake.

### 7.2 Numerical Implementation

As seen in the previous section, it is necessary to impose extra conditions at the free surface panels immediately downstream of the transom. By incorporating both the free surface boundary conditions and the continuity requirements at the stern, however, the problem becomes over-specified.
7.2.1 Transom Conditions

The linearized kinematic boundary condition for a given wave elevation and slope provides an expression for the normal velocity at the linearized free surface, \( z = 0 \).

\[
\frac{\partial \varphi}{\partial z} = \frac{\partial \zeta_T}{\partial t} - (\vec{W} - \nabla \Phi) \cdot \nabla \zeta_T - \frac{\partial^2 \Phi}{\partial z^2} \zeta_T \tag{7.5}
\]

This, however, is in conflict with the solution of the mixed boundary-value problem. Recall that the same value \( \frac{\partial \varphi}{\partial z} \), is calculated on the entire free surface by solving the boundary-integral equation (2.12). In general, the value obtained by Green's identity does not satisfy (7.5) at the stern.

Similarly, the dynamic free surface boundary condition just aft of the stern can be used to determine the value of the potential on the linearized free surface.

\[
\frac{\partial \varphi}{\partial t} - (\vec{W} - \nabla \Phi) \cdot \nabla \varphi = -g \zeta_T + \left[ \vec{W} \cdot \nabla \Phi - \frac{1}{2} \nabla \Phi \cdot \nabla \Phi \right] \tag{7.6}
\]

But for a two-dimensional transom flow, Schmidt [43] used a flat-ship linearization to show that the curvature of the free surface has a square-root singularity at the transom. The bi-quadratic spline functions which are used by the present numerical method to represent the unknowns on the free surface, cannot accommodate the infinite curvature which is necessary to impose zero pressure at the stern. This leads to numerical difficulties, as a solution cannot be found which satisfies (7.6) for a specified wave elevation.

The above difficulties are mainly due to the fact that the transom conditions are basically inconsistent with the free surface linearization. Linear theory assumes that the variation of the hull shape near the waterline is small. Then, the exact position of the intersection of the hull with the free surface has a negligible effect on the flow. But for transom flows, where the hull curvature is infinite and a particular line of detachment is desired, the above assumptions are not valid. Raven [40] found that by adopting a fully non-linear treatment of the boundary conditions for the steady flow problem, the difficulties disappeared.
7.2. Numerical Implementation

For the present implementation, there is a choice of the condition used to specify each of the three unknowns — wave elevation, potential, and normal velocity — at the free surface immediately aft of the transom. Each one may be specified by using three alternatives; continuity from the hull, a free surface boundary condition, or Green's second identity. A study was carried out to determine a combination which gave both convergent and physically plausible results.

It was concluded that the best implementation of the transom conditions was to specify the wave elevation according to (7.2), the potential by continuity from the hull, and the normal velocity through the solution of the boundary-integral equations. The wave slope was not explicitly specified according to (7.3) and (7.4) but it was found to have a realistic behavior for the practical calculations which were performed. Even if the free surface slope had been imposed, however, it would have been difficult to exactly simulate the physical behavior of the flow using bi-quadratic spline functions, because of the theoretically infinite curvature at the trailing edge.

The same approach was used for the dynamic boundary condition. Equation (7.6) was not directly enforced, but in practice the resulting non-zero pressure at the stern was found to have a very localized effect.

7.2.2 Local Flow

An additional numerical difficulty was encountered for the present time-domain method due to the division of the perturbation potential into local and memory flows. The local flow, which is the pressure release problem \((\varphi = 0\) on the free surface), separates the local added mass from the forcing in the equations of motion. The motivation for this stems from numerical stability considerations, as shown by Kring [18]. For transom flows, however, the sharp corner at the lower edge of the transom becomes the source of high velocities in the local flow, which affect the total solution.

This problem was solved by paneling the transom with a spline-sheet of specified potential and solving for the normal velocity using the boundary-integral equations. The value of the potential in the interior of the transom surface is interpolated from the hull panels at the edge of the transom. This leads to a slowly varying distribution
of potential on the transom, which ensures that the major component of velocity is in the normal direction, into the fluid. The large velocities from the corner in the local flow are hence eliminated and a smooth solution is obtained.

### 7.3 Validation

The above implementation was validated using a simple test hull, the lines of which are shown in figure 7-2. This hull has a deep transom, beam to length ratio \( B/L = 0.1 \), and draft to length ratio \( T/L = 0.0625 \). The maximum depth of the transom is 1.25% of the length of the hull. The solution was obtained at a hull Froude number of \( F_n = 0.4 \) which is equivalent to a transom depth Froude number of \( F_{nr} \approx 3.56 \).

The convergence of the wave patterns and pressure distribution on the hull was investigated for an increasing number of panels. Figure 7-3 shows the results of this convergence study for the steady flow problem, where some contour plots of the wave elevation and the pressure distribution on the hull are presented for an increasing number of panels.

The same quantities are displayed in figure 7-4, where a comparison is made...
Figure 7-3: Contour plots of the wave elevation and pressure distribution on a transom-stern hull, for an increasing number of panels.
Figure 7-4: A comparison between the present time-domain method (a) and an equivalent frequency-domain method (b) for the steady flow solution past a transom-stern hull.
Figure 7-5: The heave (a) and pitch (b) RAO of a transom-stern ship. Convergence with an increasing number of panels and comparison with a frequency-domain method.

between the steady solutions of the present time-domain method and an equivalent frequency domain method (*SWAN 1*). It must be noted that the frequency-domain method employs a different set of transom conditions, which causes the dynamic pressure at the transom edge to come closer to balancing the hydrostatic pressure, as theoretically required. Due to stability problems, it was not possible to employ these same conditions in the time-domain. The aforementioned conditions which were employed, however, result in a wave pattern and pressure distribution almost identical to the frequency-domain method, with the exception of the body in the immediate vicinity of the transom. As expected, however, this effect is localized and does not affect the global solution. It can also be argued that the time-domain solution is closer to the physical solution due to the singularity at the transom edge. The present method eliminates the singularity in the formulation of the transom conditions and thus obtains a smooth solution. The frequency-domain method, however, attempts to capture the exact value of the pressure at the stern but the spline functions cannot resolve the singular behavior of the flow in the area, thus possibly resulting in a pressure drop over a larger area of the hull than what occurs in reality. Experiments were not available to support or deny this argument.
The problem of motions in waves was examined next, and figure 7-5 displays the convergence of the heave and pitch Response Amplitude Operators with an increasing number of panels. A comparison with the frequency domain method is also shown in the same figure. After resolving the numerical difficulties in the local flow, the convergence of the method is excellent. The agreement with the frequency-domain method is fair, with the differences again being due to the different transom conditions employed by the two methods. As will be presented in chapter 10, experimental data suggest that the motions as predicted by the current time-domain method are more accurate.

7.4 Conclusions

A robust, convergent method has been developed, which is capable of predicting the flow past deep transom sterns. This problem was previously unsolved in the time-domain, because of numerical difficulties. These difficulties have been resolved by paneling of the “dry” portion of the stern, which had a smoothing effect on the local flow, and by choosing appropriate transom conditions.

It must be noted that although the present method presents satisfactory results it does not model the precise behavior of the flow at the transom edge. The local effect of infinite curvature leading to zero pressure at the stern is not captured by this linear method. In fact, the real flow near the transom is not very suitable for linearization. Fortunately this is a very local anomaly, similar to the spray near the bow, for example. The transom conditions which have been employed eliminate the singularity at the stern, thus producing a solution which is believed to be closer to the actual flow than what is obtained when the exact value of pressure is enforced at the transom.

The method is intended for flows which separate exactly at the transom edge. While this accurately models the flow behavior for most vessels with such sterns, there are some cases where it is not applicable. The flow past the transoms of IACC sailing yachts is one such non-linear example which will be examined in more detail.
in chapter 9.
Part III

Applications

1992 America’s Cup Defender Trials
This chapter presents some applications of the numerical method to conventional ships. Since more experimental data exist for such vessels than for advanced marine vehicles, these cases serve as validation of the method in order to gain more confidence when examining more complex cases.

The wave patterns are checked for convergence and are compared to experiments for two characteristic hulls. Section 8.1 presents results for a single-screw merchant ship, the Series 60 ($C_B = 0.7$). A modern naval ship, model 5415, is examined in section 8.2.

### 8.1 Series 60

The lines drawing of the Series 60 hull which is examined in this section is shown in figure 8-1. All the results presented below are for a Froude number $F_n = 0.316$.

When analyzing the flow past a ship, it is important to have confidence that the numerical solution is not affected by the truncation of the free surface. Figure 8-2 provides proof that the for a sufficiently large domain size, the wave pattern near the ship is virtually unaffected. The wave patterns for the two domain sizes shown differ only near the edges, where the effect of the damping zones becomes significant.

Figure 8-3 presents some contour plots of the wave elevation on the free surface.
Figure 8-1: The lines drawing of the Series 60.

Figure 8-2: Domain sensitivity for the Series 60.
Figure 8-3: Contour plots for the wave elevation and pressure distribution on the Series 60 hull, at Froude number 0.316, for an increasing number of panels.
Figure 8-4: The wave profile along the hull (a) and the wave heights at a cut along y/L=0.108 (b) for the Series 60 hull at Froude number 0.316. Results are shown for an increasing number of panels and are compared to experiments.

and the pressure on the body, with an increasing number of panels. The solution seems to converge, and this may be observed more easily from the plots of the wave profile along the hull and along a longitudinal cut at y/L = 0.108, as shown in figure 8-4. The numerical results not only are convergent, but also agree very closely with experiments. The slight differences, which are most noticeable immediately downstream of the stern, are attributed to viscous and other non-linear effects which are not modeled by the present method.

8.2 Model 5415

With a sonar dome and a deep transom stern, Model 5415 represents a naval combat ship with a considerably more complex form than the Series 60, examined above. The lines for this hull are shown in figure 8-5.

Experimental data were available for this hull at a Froude number of $F_n = 0.2755$. At this speed, the transom depth Froude number as defined by (7.1) is equal to 3.8, and hence flow separation is most likely to occur at the stern.

Although the wave profile along the hull and along a cut at y/L = 0.097 show
Figure 8-5: The lines drawing of Model 5415.

Figure 8-6: The wave profile along the hull (a) and the wave heights at a cut along $y/L=0.097$ (b) for Model 5415 at Froude number 0.2755. Results are shown for an increasing number of panels and are compared to experiments.
convergence with an increasing number of panels (figure 8-6), the agreement with experiments is not as good as for the Series 60. This fact is attributed to the non-linearities associated with the sonar dome and the deep transom stern, which were not present in the more linear Series 60 hull. Nevertheless, the agreement is still as satisfactory as can be expected of a linear panel method.

The convergence is also slower relative to what was observed for the Series 60 hull. Lower speeds combined with the presence of a transom flow are known to produce wave patterns dominated by shorter wavelengths, which require a greater number of panels to be fully resolved.

Convergence properties improve as the Froude number increases, as can be observed from figure 8-7, where the wave profile along the hull and along a longitudinal cut at $y/L = 0.097$ are shown for a Froude number $F_n = 0.4136$.

Even though the wave patterns for the higher Froude number converge faster, however, an area still seems to exist, aft of the deep transom stern, where rate of convergence is not as fast as would be desired. The reason for this poor behavior is due the inconsistency with linear theory of the physical flow at the outer corner of the transom. The error propagates out and aft with the flow, following the diverging
wave pattern of the ship.

It is, however, very encouraging to note that in all cases the convergence of the wave profile on the hull is excellent. This is an indication that the integrated forces are also convergent. Quantities such as wave elevation and pressure are only used over the surface of the hull for the evaluation of the wave forces.
This chapter presents a complete case study involving the IACC sailing yacht shown in figure 9-1. Sailing vessels have complex hulls and many of the extensions provided in this thesis have been used to carry out the analysis. In fact, most of the extensions were actually motivated by the study of sailing yachts.

Due to the proper treatment of the keel and rudder as lifting surfaces (chapter 5), the panel method is used to predict the interaction of those appendages with the free surface. The validity of the existing practice of treating the keel and rudder independently of the free surface flow is thus examined.

The flow near the transom of the vessel is especially complex, and requires the numerical approach developed in chapter 7. The steady resistance is hence estimated and an approximate method to treat the non-linearities in this area is devised.

Finally, a full time-domain simulation is performed by modeling the flow in air and in water simultaneously. The sails are treated according to the thin body theory developed in chapter 6 and the aerodynamic forces are coupled with the hydrodynamic forces through the equations of motion. It is therefore possible to investigate the effect of the added mass and damping of the sails as the vessel pitches and rolls. This coupled approach in the time domain allows the assessment of the forces on the rig in irregular seas, even with the future inclusion of non-linear effects in the code.
Figure 9-1: The lines drawing of an IACC yacht.
9.1 Gridding Issues

9.1.1 Asymmetry

The most striking visual difference of the underwater portion of a sailboat hull from a conventional ship is the presence of a fin-keel, which generates lift to balance the aerodynamic side-force from the sails. With the exception of any trim tabs, this keel is port-and-starboard symmetric and, therefore, the yacht needs to sail under a mean leeway angle in order to produce the required amount of lift. In addition, sailing yachts usually travel under heel in order to provide static righting moment to balance the dynamic heeling moment of the rig and keel. This presence of heel and yaw eliminates the usual advantage of using symmetry to panel only half of the domain, since the hull becomes asymmetric with respect to the mean flow direction.

The need to panel the whole domain, coupled with the presence of additional panels due to the appendages and possibly even the sails, places unusually high demands on computing power.

The solution was provided by the use of a polar grid on the free surface, which uses considerably less panels than a standard rectangular grid for the same resolution near the body. Such a paneling scheme is also a natural fit for the rounded transoms at the waterline that many sailing yachts possess. As an illustration, the polar and the equivalent rectangular grids presented in figure 9-2 may be compared.

9.1.2 Appendages

Section 5.4 has presented some results involving lift of simple bodies. When a more complex geometry such as the combination of a hull and a keel is considered, there are some additional issues that need to be resolved.

One problem is that when the keel is modeled as a lifting surface but the rest of the hull is not, then a tip vortex forms at the intersection of the two bodies, inducing large velocities on the hull. This leads to numerical difficulties. In reality, of course, the sectional lift coefficient does not fall to zero at that intersection, since the hull
Figure 9-2: A comparison of polar and rectangular free surface gridding.
Figure 9-3: The discretization of the hull and keel of an IACC yacht.

also produces a small amount of lift. The numerical solution is to extend the keel up \textit{inside} the body to the $z = 0$ plane. The wake is then extended to pass through the hull and ceases to cause numerical problems. Normal boundary and Kutta conditions are applied to the internal portion of the lifting surface, but such panels are ignored during pressure integration for force calculations. An analogous situation occurs at the intersection of the fin-keel with the bulb.

The need to extend the wake through the hull eliminates some freedom of the hull discretization. It is now necessary to have a cut in the hull spline sheet so that the wake panels do not pass through any hull panels. The simplest way to accomplish this is to discretize the hull in two sheets along the line of port-and-starboard symmetry. Then, however, the wake needs to be aligned with the plane of symmetry of the vessel and not with direction of the free stream flow. Fortunately, as shown in section 5.4, the numerical method is not sensitive to small variations in the direction of the wake.
Figure 9-4: The resistance coefficient of an IACC yacht. Convergence of the numerical results using pressure integration together with the waterline integral (a) and using simple pressure integration (b). The results by the approximate non-linear method are also shown in (b). The coefficient is based on the surface of the hull below the $z = 0$ plane.

Figure 9-3 shows the resulting discretization of the hull and keel configuration which was tested.

### 9.2 The Steady Resistance Problem

#### 9.2.1 A Non-Linear Extension

Chapter 3 has described a method of resistance calculation which is used in conjunction with the present linear method to account for the dynamic wetted area of the hull (3.6). This involves the evaluation of a waterline integral in addition to the pressure integration over the hull below the $z = 0$ plane. Strong hull non-linearities, however, are not expected to be fully captured by this method.

Sailing yachts commonly have hulls with significant flare and overhang at the bow and stern. This causes the shape of the under-water portion of the hull to change considerably with speed. But even with the highly non-linear hull of figure 9-1, the numerical results converge to a resistance curve relatively close to the experiments (figure 9-4(a)).
In some cases, however, a better degree of accuracy is desired, which is beyond the capabilities of a linear method. Recently, fully non-linear panel methods have started to become practical [40] — even if only for the steady resistance problem — but they still require considerable computational effort. An approximate non-linear extension to the present linear method was therefore developed, with the hope of capturing the non-linearities associated with the dynamic wetted surface, without significantly increasing the computational intensity.

The method consists of performing an initial run in order to find the intersection of the hull with the free surface. The shape of the dynamic wetted surface of the hull is then estimated, taking into account any sinkage and trim. A new hull-form is then produced by vertically shifting each section so that the intersection with the free surface moves to the $z = 0$ plane. A new run is performed with this modified hull. The wave resistance is calculated by integration of the normal pressure force, using the normal vectors of the original hull at any given $x-y$ position. The force is then scaled according to the ratio of the displacement of the original to the "stretched" hull. This is necessary because, in general, the displacement of the modified hull is found to be greater than that of the original one, a phenomenon which does not occur in reality since the vessel has a fixed weight.

This author recognizes the crude engineering nature of the above approach to this problem. No further attempt will, therefore, be made to justify the theoretical basis of the method, other than to note its exceptional agreement with the experimental results, shown in figure 9-4(b). The simple pressure integration consistently over-predicts the wave resistance of the vessel. The subtraction of the waterline term (3.6) brings the curve closer to experiments, but seems to miss the exact trend at low speeds. As expected, the non-linear approach has the same general effect as the waterline term, but is much more accurate. The difference with experiments at the high end of the speed range is due to non-linearities in the free surface condition, which are not modeled in either method.
9.2.2 Appendage–Free Surface Interaction

When testing sailboat appendage configurations, usually in wind tunnel facilities, it is common to ignore the effect of the free surface. The justification is that the appendages are far enough immersed that the free surface may be treated as a rigid wall. Conversely, hulls are often tested in towing tanks without the proper appendages in place, assuming that their effect can be considered separately.

The numerical tool that has been developed, now enables the simultaneous study of the hull, the keel, and the free surface. Figure 9-5, for example, compares the free surface elevation and the pressure on the body for the hull with and without the keel in place. It can be easily seen that the presence of the keel strongly affects the flow near the free surface, especially on the windward side (right side as shown in figure 9-5) of the yacht. Reciprocally, the free surface is expected to affect the flow near the keel. This interaction suggests that caution needs to be exercised before isolating the appendages from the hull for experimental and numerical analysis.
9.3. Free Motion Simulations in Waves

9.3.1 Head Seas

The prediction of the seakeeping performance of a vessel can be just as important as the problem of steady translation. Experiments were available from the \textit{PACT'95} model testing program for the motions of the IACC yacht of figure 9-1. It was, therefore, possible to validate the enhancements to the method which have been developed as part of this thesis, as applied to the particular geometric complexities of this IACC hull. The above hull features a rounded transom stern which is characterized by a slightly different flow behavior than the deep transom case which was examined in chapter 7. Nevertheless, as can be seen in figure 9-6, the heave and pitch response amplitude operators are clearly convergent with an increasing number of panels.

Figure 9-6 also presents a comparison of the experimental versus the numerical response amplitude operators. Results using the frequency domain method, \textit{SWAN 1}, are also shown in the same figure. Satisfactory agreement with experiments is observed. In fact, the present time domain method appears to be a slight improvement over the frequency domain method.

Figure 9-6: The Heave (a) and Pitch (b) RAO for an IACC hull traveling at Froude number 0.347 in head waves.
9.3.2 Appendage Modeling

In addition to the steady motion case examined in section 9.2.2, the sailboat keel and rudder are also expected to affect the problem in the presence of waves. This is especially true for the transverse modes of motion. The present section examines the importance of accurately modeling such appendages, as opposed to treating them in an approximate fashion using lifting line theory.

The response amplitude operators were obtained for the vessel of figure 9-1 in bow seas, at an angle of incidence equal to 45 degrees from the beam. A mean heel and leeway angle were imposed in order to simulate actual sailing conditions.

In an attempt to capture the effect of the presence of the fin-keel, this component was first entirely omitted, but the bulb was taken into account by placing panels on its surface.

The fin-keel was then modeled by lifting line theory, ignoring its hydrodynamic effect on the rest of the flow. The keel lift force, which was used to modify the equations of motion, was computed according to its area, aspect ratio, speed, and angle of attack. The apparent angle of attack was adjusted according to the motions of the vessel and the incident flow velocity.

The RAOs obtained for the above appendage models were compared with the more accurate method for treating lifting surfaces which has been developed in chapter 5. The keel and bulb were paneled, and a trailing vortex sheet was added, as shown in figure 9-3.

Due to computing power constraints, the effect of the rudder was taken into account using the approximate lifting line model as described above for the keel.

Figure 9-7 presents the effect of the fin-keel on the heave, pitch, roll, and yaw motions of the sailing yacht in bow seas. It is immediately obvious that the presence of the keel significantly affects the transverse modes of motion of the yacht. The additional damping of the appendage considerably lowers the roll and yaw motion amplitudes. Although the effect on roll is well captured by the lifting line model, the yaw motions are somewhat different. Nevertheless, it can be concluded that the
Figure 9-7: The oblique wave heave (a), pitch (b), roll (c), and yaw (d) RAO for an IACC yacht under heel and leeway. Results are presented for no fin-keel, for the hull with the fin-keel modeled using lifting line theory, and for the hull with the fin-keel paneled and treated according to chapter 5.
panelization of the appendages of an IACC yacht is not necessary in the preliminary stages of the design, as the lifting line model provides satisfactory results. But if the performance is to be predicted within a few percent, the method developed in chapter 5 becomes an essential tool of evaluation.

9.4 Coupling of Aerodynamic and Hydrodynamic Flows

With the thin body theory developed in chapter 6, it is possible to model the flow of the wind past the sails of a yacht. A boundary-value problem is set up in air and solved simultaneously with the flow in water. A direct time-domain simulation for the entire vessel is therefore performed.

In order to ease the computational effort, the aerodynamic and hydrodynamic flows are coupled only through the motions of the yacht. Hence, no interaction is allowed directly through the movement of the free surface, which is treated as a rigid wall for the aerodynamic flow problem.

As an example, an IACC yacht beating to windward is considered. The vessel is heeled over, traveling at 9 knots in bow seas (45 degrees from the beam), with 4 degrees leeway angle. The apparent wind velocity is 20 knots coming from a direction of 30 degrees off the direction of motion. This configuration is shown in figure 9-8.

Both the mainsail and the genoa were modeled as thin lifting surfaces with a wake shed downstream, and the hull and keel configuration were as shown in figure 9-3. The keel was paneled and treated as a lifting surface, but the rudder was modeled using lifting line theory.

Figure 9-9 presents the oblique wave RAO for the yacht with and without the sails. The motions of the bare hull are also shown for comparison.

Totally neglecting the presence of the keel gives a very poor estimate of the transverse motions. It is, therefore, evident that the appendages of a sailing vessel need to be considered in conjunction with the hull in order to obtain an accurate evaluation
Figure 9-8: A full simulation of an IACC yacht beating to windward in waves.
Figure 9-9: The effect of the sails on the oblique wave heave (a), pitch (b), roll (c), and yaw (d) RAO for an IACC yacht under heel and leeway. The apparent wind velocity is 20 knots from a direction of 30° off the direction of motion. The yacht is traveling at a speed of 9 knots, through waves incident at 45 degrees from the beam.
of its performance in waves.

The main effect of the sails is an extra added mass with resonance at very low frequencies. This causes an amplification of the motions at long wavelengths. The small extra damping slightly reduces the roll motions at medium wavelengths, but not as much as initially expected.
Advanced marine vehicles such as semi-displacement ships, catamarans, SWATH, and SES possess several characteristics which are quite different than conventional ships and present new technical challenges. For this reason, the experimental data available for such vessels are limited and numerical modeling techniques are of increased value to the designer. The new technical challenges of this class of ships include higher speeds of operation as well as geometric complexities such as transom sterns and multiple hulls.

In addition, many high-speed vehicles are equipped with hydrofoils or motion control fins which produce lift. These features may be handled in a similar manner as what has been already described in chapter 5.

In this chapter, numerical results are presented for a semi-displacement ship in section 10.1, and for a catamaran in section 10.2. Both types of vessels are characterized by some of the aforementioned geometric complexities that are now able to be treated by the panel method.

10.1 Semi-Displacement Ship

Semi-displacement ships, which are characterized by wide shallow transoms, can operate efficiently past the limiting hull speed for displacement ships. The sharp transom
Figure 10-1: The calm-water wave pattern of the TGC770 FastShip at 40 knots.

stern forces flow detachment, thus avoiding the sharp increase in wave resistance with speed which occurs for conventional ships.

To illustrate the application of the method to a semi-displacement hull, the TGC770 Fast-Ship (Thornycroft, Giles & Company, Inc) is considered. This 229m container-ship has been designed by TGC and FastShip Atlantic, Inc. to operate across the atlantic at 40 knot speeds.

10.1.1 Steady Wave Resistance

The wave pattern of the ship traveling at 40 knots in calm water, as predicted by SWAN is presented in figure 10-1. The flow appears quite realistic, with a pronounced “rooster tail” forming behind the transom stern.

Some experimental results for the steady resistance problem have been obtained from SSPA through the courtesy of TGC and Fastship Atlantic. These results are shown in figure 10-2 along with the predictions of SWAN.
10.1.2 Motions in Head Seas

The unsteady problem of motion in regular head seas is investigated next. Branner and Sangberg [2] have presented experimental results for this vessel which they have compared to the predictions of the frequency-domain version of SWAN.

The transom conditions which are applied to the frequency-domain method are slightly different, and hence the results differ from the present time-domain version, as shown in figure 10-3. The reason for this difference is that it was not possible to reproduce the exact same conditions in the time domain due to numerical difficulties. In addition, the extra panel sheet of specified potential on the dry portion of the transom is not present in the frequency domain method. The results, however, appear to be closer to experiments in the case of the time-domain method which has been developed as part of this thesis.

From the above analysis it should be clear that realistic semi-displacement hulls can be treated effectively. The numerical difficulties presented by the transom stern in the time-domain have been overcome and reliable results can be obtained using the present method.
10.2 Catamaran

Catamarans are a very common choice of vessel at high operating speeds because their slender demi-hulls present a relatively small residuary resistance, which is the dominant component of resistance at such speeds.

This advantage often disappears at low speeds because of the interaction between the hulls. The designer is thus faced with the challenge of determining the cross-over point where a catamaran is more efficient than a monohull. This can be an extremely difficult task unless a systematic method is available to account for this hull interaction.

The interaction between the demi-hulls causes the flow to be asymmetric with respect to each hull’s plane of symmetry. The numerical treatment of such vessels is therefore similar to sailing yachts in that there is asymmetry and sideforce associated with the flow past each hull.

Chapter 5 has presented an extension of the Rankine panel method to include lifting surfaces. This approach is necessary when dealing with multi-hulled vessels, especially for hulls with sharp vertical sterns, where a Kutta condition must be enforced to ensure potential flow detachment at the same location as if viscosity were present.
10.2. Catamaran

In addition, the great majority of high speed catamarans have deep transom sterns, which need to be treated as described in chapter 7.

### 10.2.1 Lewis Form Catamaran

The method was first validated by comparing results to experimental data. The hull chosen for this was the Lewis form catamaran defined by Kashawagi [17], with a demi-hull beam to length ratio $B/L = 6$, and a separation ratio between centerlines of the demi-hulls of $S/B = 2$. Experimental results and strip theory predictions of the added mass and damping coefficients in heave and pitch are available for a forward speed of $F_n = 0.3$. Figures 10-4, 10-5, and 10-6 compare these data to the predictions given by the present method. It can be seen that the agreement of the Rankine panel method with experiments is very good, confirming the accurate modeling of the three-dimensional wave interactions between the hulls. Observing the same figure, it is evident that these interactions are not captured by strip theory.
Figure 10-5: Heave–pitch added mass and damping coefficient predictions compared to experiment for a Lewis form catamaran at $F_n = 0.3$

Figure 10-6: Pitch–pitch added mass and damping coefficient predictions compared to experiment for a Lewis form catamaran at $F_n = 0.3$
Figure 10-7: Wave resistance coefficient as a function of speed for a high-speed catamaran with various demi-hull separation ratios.

### 10.2.2 Demi-hull Interaction

The extent of the interaction between demi-hulls was next examined for a typical high-speed catamaran. A representative such vessel was designed, which was characterized by a sharp vertical stern, a flared bow, significant parallel midbody, a length to demi-hull beam ratio of $L/B = 10$, and draft to beam ratio of $T/B = 5$. A base monohull and catamarans with this demi-hull were created with various centerline separation to beam ratios, $S/B = 2, 3, 4, 8$, in order to examine the behavior of typical catamarans in calm water and in waves as a function of speed and demi-hull spacing.

First, these hulls were run in calm water conditions. If the interaction effects were small, then the resistance coefficient as a function of demi-hull separation, for example, would be approximately constant. This is not the case, however, as can be seen in figure 10-7, where the wave resistance curves as calculated by the present method are shown for several demi-hull separation ratios. There is a definite trend of significant resistance reduction with separation, over the whole range of speeds considered.

The separation of the hulls also affects the motion of the vessel in waves. The roll motion Response Amplitude Operators is recorded in figure 10-8 for the above
representative catamarans with two separation ratios at $F_n = 0.8$ in beam seas. The difference observed between the two curves is not due to the hydrodynamic interaction between the hulls. It is mainly due to the fact that, for beam seas, the roll exciting force has a minimum when the wavelength is an integer multiple of the separation.

The hydrodynamic interaction between the hulls is more clearly visible by observing the motions in head seas, shown in figure 10-9. The difference in heave and pitch is clear for various separation ratios and speeds. The proximity of the demi-hulls and their interaction causes a de-tuning effect in head waves, which decreases the amplitude of the motions at resonance.

### 10.2.3 Wave Patterns

The trailing vortex sheet that is used to model the wake behind the catamaran, restricts the flow so that the velocity at the trailing edge of each demi-hull is finite. This can be verified by the smooth wave patterns of figure 10-10, shown for Froude numbers of 0.3, 0.55, and 0.8. This would not have been the case, if the demi-hulls had not been modeled as lifting surfaces by the numerical method. The flow would then have to turn the corner of the the sharp stern which would lead to an unphysical
10.2. Catamaran

Figure 10-9: Heave and pitch RA0s for a catamaran at various separation ratios and speeds.

solution and possible numerical difficulties.

The reader may note how the wave patterns for all speeds fall within the Kelvin sector, but show a clear transfer of energy from transverse to divergent waves as the speed increases. The prediction of wave patterns is important in the design of the box structure connecting the hulls.

10.2.4 Non-linear Motions

Linear theory considers only the portion of the hull which is below the $z = 0$ plane. In some cases, however, non-linearity may be important for the prediction of ship motions. A first check to determine whether the immersion of the the hull above the still waterline is important, is to include the effects of the Froude-Krylov forces acting on that portion of the hull. This adds little extra computational effort but can be very useful in obtaining a first approximation of the effect of overhang. A precise evaluation could then be carried out using a more sophisticated non-linear method such as the weak scatterer hypothesis extension of SWAN [13]. Figure 10-11 illustrates one case where non-linear effects are strong for the motion of the representative catamaran in head seas. The figure shows a series of snapshots, where the wave pattern and
Figure 10-10: Steady wave patterns for a catamaran from moderate to high speeds.
Figure 10-11: Snapshots of the submerged hull surface for a catamaran at $F_u = 0.3$ in a regular head sea; and the corresponding time record for the linear and non-linear heave motions.
the discretization of the hull wetted surface is viewed from below and from the side. In the second snapshot, the upper cross-structure can be seen to submerge, thereby affecting the motions. This can be verified by observing the time history of the heave motion which is shown at the bottom of the same figure. The difference between the linear and non-linear simulation is by no means negligible in this case.
Concluding Comments

A robust numerical method has been developed to the point where it can be considered to be a powerful and accurate design tool for ship designers. The steady resistance, wave patterns, motions, added resistance, and loads, are all examples of useful quantities that can be obtained.

The present work has added the capability to predict free surface steady or unsteady flows past lifting surfaces, infinitesimally thin bodies, and deep transom sterns, extensions which have significantly broadened the range of applications of the method. Viscous effects and their interaction with the wave flow may now also be taken into account, resulting in a more complete picture of the total resistance of the ship.

The theoretical basis and numerical implementation for the above features has been presented, and then validated for some test cases through comparison with analytical results and experiments. Convergence of the results with an increasing number of panels has been established.

As an example of the wide range of complex hull forms that may be treated, the method was used to analyze a semi-displacement ship, a high speed catamaran, and a sailing yacht. The results were compared to experiments whenever possible, and a very satisfactory agreement was obtained. For the yacht, an approximate non-linear method was developed to account for the change of the underwater portion of the hull with speed. The interaction of the appendages with the free surface was also examined. The sails were modeled as infinitesimally thin lifting surfaces and their effect on the free motions of the vessel was predicted by performing a simultaneous time-domain simulation of the flow in air and water. The heave, pitch, roll, and yaw motions were therefore obtained for the heeled, yawed yacht beating to windward in oblique seas. In the area of advanced marine vehicles, the effect of the deep transom of a semi-displacement ship on heave and pitch was accurately predicted. The interaction of the demi-hulls of a catamaran was also investigated in calm water, head seas, and beam seas.
The time-domain approach to the problem of ship motions enables a straightforward extension to include non-linear effects. Indeed, the method has been already extended to include some non-linearities such as a viscous roll damping model and the non-linear Froude-Krylov forces in the equations of motion. In addition, stronger non-linear effects have been incorporated by the parallel development of the method using the “weak scatterer” hypothesis, a formulation which enforces the exact body boundary conditions and linearizes the free surface conditions about a non-linear incident wave. Work is also being done in applying the underlying method to solve the flow while retaining second and third order terms in the free surface boundary conditions.

The efficiency of the numerical method needs to be improved before the solution of non-linear problems becomes practical. With computing power becoming increasingly available, even the fully non-linear problem could be conceivably tackled in the future. Prior to that, however, it was necessary to obtain further experience and understanding of the linear numerical solution to practical problems. The present work has significantly contributed to this area.
APPENDIX A

THE BOUNDARY LAYER MODEL

A.1 Definition of Main Parameters

Displacement thickness,
\[ \delta^* = \int_0^\infty \left(1 - \frac{U}{U_e}\right) dy \] (A.1)

Momentum thickness,
\[ \theta = \int_0^\infty \frac{U}{U_e} \left(1 - \frac{U}{U_e}\right) dy \] (A.2)

Shape parameters,
\[ H = \frac{\delta^*}{\theta} \] (A.3)
\[ H_1 = \frac{1}{\theta} \int_0^\delta \frac{U}{U_e} dy = \frac{\delta - \delta^*}{\theta} \] (A.4)

Skin-friction coefficient,
\[ C_f = \frac{\tau_w}{\frac{1}{2} \rho U_e^2} \] (A.5)

Entrainment coefficient,
\[ C_E = \frac{1}{r U_e} \frac{d}{dx} \left(r \int_0^\delta U dy\right) = \frac{1}{r U_e} \frac{d}{dx} (r U_e H_1 \theta) \] (A.6)
A.2 Flow over Solid Surfaces

Boundary layer development, specified by the three independent parameters $\theta$, $H$ and $C_E$, is predicted by the numerical integration of the three simultaneous ordinary differential equations:

\[
\frac{d}{dx}(r\theta) = \frac{rC_f}{2} - (H + 2)\frac{r\theta}{U_e} \frac{dU_e}{dx} \tag{A.7}
\]

\[
\frac{\theta dH}{dx} = \frac{dH}{dH_1} \left[ C_E - H_1 \left\{ \frac{C_f}{2} - (H + 1)\frac{\theta}{U_e} \frac{dU_e}{dx} \right\} \right] \tag{A.8}
\]

\[
\frac{\theta dC_E}{dx} = F \left[ \frac{2.8}{H + H_1} \left\{ (C_f)^{1/2} - \lambda C_f^{1/2} \right\} + \left( \frac{\theta}{U_e} \frac{dU_e}{dx} \right)_{EQ} - \frac{\theta}{U_e} \frac{dU_e}{dx} \right] \tag{A.9}
\]

In these equations, $r$ is the body radius in axisymmetric flow, set to unity for two-dimensional flow. The various dependent variables and functions are evaluated from the following relationships:

For $C_f$: from the known surface-pressure distribution the local free-stream velocity, $U_e$, is evaluated from gas-dynamic relations and the absolute viscosity $\mu$ from an appropriate relation such as Sutherland’s.

Then we write:

\[
R_\theta = \frac{\rho U_e \theta}{\mu} \tag{A.10}
\]

\[
C_{f_0} = \frac{0.01013}{\log_{10}(R_\theta) - 1.02} - 0.00075 \tag{A.11}
\]

\[
1 - \frac{1}{H_0} = 6.55 \sqrt{\frac{C_{f_0}}{2}} \tag{A.12}
\]

\[
C_f = C_{f_0} \left\{ 0.9 \left( \frac{H}{H_0} - 0.4 \right)^{-1} - 0.5 \right\} \tag{A.13}
\]
For \( H_1 \) and \( dH/dH_1 \):

\[
H_1 = 3.15 + \frac{1.72}{H - 1} - 0.01(H - 1)^2
\]

(A.14)

\[
\frac{dH}{dH_1} = -\frac{(H - 1)^2}{1.72 + 0.02(H - 1)^3}
\]

(A.15)

For \( C_\tau \) and \( F \):

\[
C_\tau = 0.024C_E + 1.2C_E^2 + 0.32C_{f_0}
\]

(A.16)

\[
F = \frac{0.02C_E + C_E^2 + 0.8C_{f_0}/3}{0.01 + C_E}
\]

(A.17)

For secondary influences:

If \( R \) is the radius of longitudinal curvature, positive on a convex wall, we write

\[
R_i = \frac{2}{3} \frac{\theta}{R} (H + H_1) \left( \frac{H_1}{H} + 0.3 \right)
\]

(A.18)

\[
\beta = 7 \quad \text{for} \quad R_i > 0, \quad \beta = 4.5 \quad \text{for} \quad R_i < 0
\]

(A.19)

\[
\lambda_1 = 1 + \beta R_i
\]

(A.20)

\[
\lambda_2 = 1 - \frac{7}{3} \left( \frac{H_1}{H} + 0.3 \right) \left( H + H_1 \right) \frac{\theta}{r} \frac{dr}{dx}
\]

(A.21)

\[
\lambda = \lambda_1 \lambda_2
\]

(A.22)

It should be borne in mind that these correction formulae are of a provisional nature, and are best programmed as an optional subroutine. As they are believed to be justified only when secondary influences are small to moderate, the following limit is
arbitrarily imposed

\[ 0.4 < \lambda < 2.5 \]

in the computer program, with any value of \( \lambda \) outside either of these limits reset equal to the limit.

For equilibrium quantities:

\[
\left( \frac{\theta \, dU_e}{U_e \, dx} \right)_{EQo} = \frac{1.25}{H} \left\{ \frac{C_f}{2} - \left( \frac{H - 1}{6.432H} \right)^2 \right\} \quad (A.23)
\]

\[
(C_E)_{EQo} = H_1 \left\{ \frac{C_f}{2} - (H + 1) \left( \frac{\theta \, dU_e}{U_e \, dx} \right)_{EQo} \right\} \quad (A.24)
\]

and

\[
(C_T)_{EQo} = 0.024(C_E)_{EQo} + 1.2(C_E)_{EQo}^2 + 0.32C_{fo} \quad (A.25)
\]

Also, writing

\[
C = (0.024(C_E)_{EQo} + 1.2(C_E)_{EQo}^2 + 0.32C_{fo})\lambda^{-2} - 0.32C_{fo} =
\]

\[
= (C_T)_{EQo} \lambda^{-2} - 0.32C_{fo} \quad (A.26)
\]

we have

\[
(C_E)_{EQ} = \sqrt{C/1.2 + 0.0001 - 0.01} \quad (A.27)
\]

whence

\[
\left( \frac{\theta \, dU_e}{U_e \, dx} \right)_{EQ} = \left( \frac{C_f}{2} - \frac{(C_E)_{EQ}}{H_1} \right) / (H + 1) \quad (A.28)
\]

Equations A.10 to A.28, provide the dependent variables needed to evaluate equations A.7, A.8 and A.9 at each stage of the numerical integration.

### A.3 Wake Flows

To continue a boundary layer calculation past a trailing edge, so that one side of the wake is calculated at a time, for \( x > x_{TE} \):
1. by-pass equations A.10 to A.13 and set

\[ C_f = C_{f_0} = 0 \quad (A.29) \]

2. replace equation A.22 by

\[ \lambda = \frac{1}{2} \lambda_1 \lambda_2 \quad (A.30) \]

### A.4 Boundary Conditions

To specify the problem, free-stream stagnation properties \( p_0 \) and \( T_0 \) and streamwise distributions of \( r(x) \), or some equivalent information must be given. Sometimes longitudinal curvature of the surface \( R(x) \) will also be required. Ideally, initial values of \( \theta, H \) and \( C_E \) should also be given, but these will usually be known only when making comparisons to experiment. The minimum practicable information is an initial value of \( \theta \); frequently calculations are started at an assumed transition point, and the initial \( \theta \) is estimated by applying a simple, approximate method to the laminar boundary-layer development from its origin to the transition point. Initial \( H \) may then be estimated either from equation A.12, i.e. assuming a flat-plate velocity profile at the starting point, or else by assuming the flow is locally in equilibrium and using equations A.23 to A.28 to evaluate \( H \) given \( (\theta/U_e)(dU_e/dx) \). In this case, it is simplest to assume \( \lambda = 1 \) and determine \( H \) from equation A.23, but even to do this a process of iteration is required. With \( \theta \) and \( H \) known, the initial value of \( C_E \) may be taken as its equilibrium value. In this case, whether or not we assume \( \lambda = 1, C_E \) may be determined directly as \( (C_E)_{EQ} \) from equation A.27 (if \( \lambda = 1 \), equation A.24 will do) and the preceding equations.
Bibliography


