Advances in Ducted Propulsor Analysis using Vortex-Lattice Lifting-Surface Techniques

by

Gerard Paul Mc Hugh

B.A., Mathematics, University of Dublin, Trinity College (1995)
B.A.I., Mechanical Engineering, University of Dublin, Trinity College (1995)

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

Naval Engineer

at the

MASSACHUSETTTS INSTITUTE OF TECHNOLOGY

September 1997

© Massachusetts Institute of Technology 1997. All rights reserved.

Author. Department of Ocean Engineering
August 8, 1997

Certified by. Justin E. Kerwin
Professor of Naval Architecture
Thesis Supervisor

Accepted by. J. Kim Vandiver
Chairman, Departmental Committee on Graduate Students
Advances in Ducted Propulsor Analysis using Vortex-Lattice Lifting-Surface Techniques

by

Gerard Paul Mc Hugh

Submitted to the Department of Ocean Engineering
on August 8, 1997, in partial fulfillment of the
requirements for the degree of
Naval Engineer

Abstract

Current trends in the design of underwater vehicles include the use of single- and multi-stage ducted propulsors combined with a highly tapered after-body. While a Vortex-Lattice lifting-surface design and analysis code has been in existence for a number of years, this potential flow representation has not included the effects of the flow through the tip-gap region i.e. between the blade tip and duct. The resulting potential flow solution in this region is not an accurate representation of the physical behavior. This deficiency in the current model has motivated the implementation of a more accurate, semi-empirical, tip-gap flow model based on an orifice flow formulation.

As an initial step in implementing the tip-gap flow model in the existing code, the method of modelling the hub and duct was revised; whereas previously vortex-lattice and source element images were used to model these surfaces, an improved method was implemented whereby the Kinematic Boundary Condition (zero flux through the boundary) is imposed directly on the hub and duct vortex-lattice surfaces, in much the same way as the blade mean camber surface is represented.

Implementation of the tip-gap flow model is based on a "Porous Foil" method; The flow in the gap region is specified according to an empirical orifice equation, relating the gap velocity to the pressure jump across the blade. An iterative scheme is used to solve for the circulation distribution on the blade, hub and duct, given the nonlinear nature of the system once the gap vortex-lattice elements and control points have been introduced. Once the circulation distribution on the blade has been determined the induced velocity on the blade, and hence the thrust and torque, can be evaluated in the usual manner (i.e. using Kutta-Joukowski’s Law).

Validation studies of the existing Coupled Viscous Propeller Blade Design and Analysis technique have been performed using experimental data from two cases:

1. HIReP large-scale two-stage turbomachine pump (stator-rotor combination)- essentially an internal flow case.


Thesis Supervisor: Justin E. Kerwin
Title: Professor of Naval Architecture
Acknowledgments

"Come to the edge," he said.
They said, we are afraid.
Come to the edge, he said.
They came.
He pushed them ... and they flew."

Guillaume Apollinaire

During my time at MIT thus far, Professor Justin Kerwin has provided the "push" to enable me to "fly" - a gentle "push" that comes in the form of constant, good-humored support, encouragement and advice - both academic and non-academic. More importantly, I am grateful that, through him, I have been afforded the flexibility to pursue an "unconventional" career path - a path along which I might otherwise have encountered more obstacles! Thank you!

Also, the "Propnuts" group at MIT has provided an exciting working atmosphere. I thank them all, but special thanks go to Todd Taylor and Scott Black for their support and encouragement during the "dark hours" of propeller code design and development.

Although not directly connected with my thesis research, Professor Mark Welsh (LCDR-USN) provided clear, gentle guidance for a large portion of my Naval Engineer's Degree program, in addition to providing advice and support regarding career direction. Through him, and the Naval Construction and Engineering Program, I have been able to pursue a broad curriculum in Naval Architecture. Again, a sincere thank you!

Finally, I thank all of the above for helping me to remain focused on my ultimate career objectives - and for helping me to maintain a happy balance between work and play!

Support for this research was provided by the Office of Naval Research; Grant # N00014-96-1-0389; Dr. Edwin Rood, Program Manager.
Contents

1 Introduction ............................................................................................................... 13

1.1 Overview ............................................................................................................. 13

1.2 Background .......................................................................................................... 16

1.2.1 Marine Propulsor Design and Analysis Tools ................................................. 16

1.2.2 Hub and Duct Modeling ................................................................................. 16

1.2.3 Tip-gap Flow Modeling ................................................................................. 17

1.3 Validation Cases ................................................................................................. 17

2 Coupled Lifting-Surface Design / Analysis Methodology ..................................... 18

2.1 Overview ............................................................................................................. 18

2.2 Vortex Lattice Method ....................................................................................... 18

2.2.1 Propulsor Design and Analysis ................................................................ 19

2.2.2 Solution Procedure ....................................................................................... 20

2.3 Propulsor Inflow ............................................................................................... 20

2.4 Viscous Flow Solver .......................................................................................... 22

2.5 Coupled Lifting Surface Design / Analysis Methodology .................................. 22

2.5.1 Descriptive Overview .................................................................................. 23

2.5.2 Mathematical Formulation .......................................................................... 24

3 Potential Flow Modeling of Hub and Duct .......................................................... 28

3.1 Overview ............................................................................................................. 28

3.2 Method of Images .............................................................................................. 29

3.3 Vortex Lattice Hub / Duct ............................................................................... 30

3.3.1 Hub and Duct Geometry ............................................................................. 30

3.3.2 Transition Wake Geometry .......................................................................... 32

3.4 Hub / Duct Circumferential Spacing ................................................................. 32

3.5 Computational Aspects of Hub, Duct Modeling .............................................. 35

3.5.1 Method of Images ....................................................................................... 35

3.5.2 Vortex Lattice Hub and Duct .................................................................... 35

3.6 Verification of Solution Procedure for Vortex Lattice Hub and Duct ............... 36

3.7 Role of Hub and Duct in Coupled Analysis ...................................................... 36

3.8 Convergence Studies ......................................................................................... 38
4 Numerical Implementation of the Tip-gap Flow Model

4.1 Overview ..................................................... 41
4.2 Viscous Flow in the Tip-Clearance Region .................. 41
4.3 Porous Foil Model .......................................... 42
4.4 Integration of the Tip-gap Flow Model in PBD-14 .......... 43
   4.4.1 Blade lattice Modification ............................ 43
   4.4.2 Adjustments to Solution Procedure .................. 43
   4.4.3 Iterative Solution of the Flow in the Tip-gap .... 44
4.5 Previous Tip-gap Treatment in PBD-14 .................... 44
4.6 Tip-gap effects! ........................................... 44

5 Case I: HIREP ................................................. 47

5.1 Overview .................................................... 47
5.2 Experimental Facility ..................................... 47
5.3 Rotor Thrust and Torque ................................ 48
5.4 Velocity Measurements ................................... 49
   5.4.1 Reference Velocity ................................. 49
   5.4.2 Velocity Profiles ................................. 50
5.5 Tip-gap Flow ................................................ 51
5.6 Overview: Coupled Viscous / Potential Flow Analysis .... 52
   5.6.1 Flow Domain Representation ....................... 52
   5.6.2 Grid Generation .................................... 53
5.7 Vortex Lattice Hub and Duct .............................. 55
5.8 Results and Discussion ................................... 56
   5.8.1 Rotor Thrust and Torque ......................... 56
   5.8.2 Velocity Profiles ................................. 58
   5.8.3 Tip-gap Flow Modeling ......................... 61
5.9 Summary .................................................... 61

6 Case II: NSMB Ka4-55 Ducted Propulsor ..................... 62

6.1 Overview .................................................... 62
6.2 Hull Problem: Setup ...................................... 63
   6.2.1 Grid Generation .................................... 63
   6.2.2 RANS Converged Solution ....................... 64
6.3 Propeller Problem: Setup ................................ 64
6.4 Coupled Analysis ......................................... 65
6.5 Discussion of Results ................................... 66
   6.5.1 Rotor Thrust and Torque ....................... 66
   6.5.2 Tip-gap Flow Modeling ......................... 67
6.6 Summary .................................................... 68
7 Conclusion

7.1 Vortex Lattice Lifting Surface Code Enhancements .......................... 70
  7.1.1 Vortex Lattice Hub and Duct ........................................ 70
  7.1.2 Numerical Tip-gap Flow Modeling .................................... 71
7.2 Concurrent Work on *PBD-14* ................................................. 71
7.3 Future Vortex-Lattice Enhancements ......................................... 71
  7.3.1 Vortex Lattice Hub / Duct in Design Mode ............................ 71
  7.3.2 Blade Wake Alignment .................................................. 72
  7.3.3 Tip-fin Modeling ....................................................... 72
7.4 Internal Flows ........................................................................ 72
7.5 Coupled Analysis : Closing Remarks ....................................... 73

A Hub Pressure Correction .......................................................... 74
  A.1 Review of Derivation ......................................................... 74
  A.2 *HIREP* Correction factor .................................................. 76
  A.3 *HIREP* Experimental Values of $K_t, K_q$ ............................. 77

B Additional Validation Cases ....................................................... 78
  B.1 Overview ............................................................................ 78
  B.2 Propeller 4119 ..................................................................... 79
    B.2.1 Features ......................................................................... 79
  B.3 Propeller 4497 / MIT-PRESWIRL ......................................... 82
    B.3.1 Features ......................................................................... 82
  B.4 Propeller R4-55 ..................................................................... 85
    B.4.1 Features ......................................................................... 85
# List of Figures

1-1 Ducted Propulsor - the duct enclosing the propeller is often referred to as a shroud or nozzle. ........................................... 14

1-2 *K4-55* Ducted Propulsor, similar to that shown in Figure 1-1. A two-dimensional view of the blade and duct section, illustrating the tip-gap, is shown in the inset figure. ............................... 15

2-1 Macroscopic view of *RANS* - potential flow propeller design / analysis coupling. The process revolves around the interchange of flow fields and forces between processes .................................... 23

2-2 Flowchart of steps involved in *RANS* - potential flow propeller design / analysis coupling ............................................. 24

2-3 Viscous Flow Solver coupling ........................................ 25

3-1 2D Hub Image. Placement of point vortex and image create streamlines at radius = 0.5 .................................................. 29

3-2 *PBD* Hub and Duct Images ........................................ 30

3-3 Hub vortex lattice in blade region ................................... 31

3-4 Vortex Lattice Representation of Hub and Duct. The propulsor shown is the MIT Sirenen case. ....................................... 32

3-5 Hub, Duct transition Wake ........................................... 33

3-6 Total velocity at the Hub and Blade control points. The flow is tangent to the surface, thus illustrating the imposed kinematic boundary condition of zero flux through the surface .......................... 36

4-1 Tip-gap Vortex Lattice and Control Points ......................... 43

4-2 Schematic representation of iterative solution to flow in tip-gap region ................................................................. 45

4-3 Effects of varying tip-gap dimension on Spanwise Distribution of Circulation. The viscous nature of the flow in the clearance region has *NOT* been considered. (Note: gap dimensions stated as percentage of rotor diameter) .................................................. 46
4-4 Effect of Tip-gap on Spanwise Distribution of Circulation. These results were derived from the Vortex lattice hub and duct implementation of PBD-14 using the tip-gap flow model (Orifice Coefficient: $C_Q = 0.84$) ........................................ 46

5-1 Hirep Rotor and Stator Geometry ................................ 48
5-2 Cross section through HIREP water-tunnel facility ............... 49
5-3 Axial, radial and Tangential velocity measured at a plane 37% upstream of the Inlet Guide Vanes .......................... 50
5-4 Locations of experimental velocity profiles .......................... 51
5-5 HIREP equivalent annular pipe representation. As the RANS formulation is axisymmetric only the upper zones are modeled numerically - the other zones are shown to indicate the full pipe domain. ........... 53
5-6 HIREP grid ................................................................ 54
5-7 HIREP rotor vortex lattice hub and duct. Note blended circumferential spacing (portion of duct omitted for clarity) .................. 55
5-8 Coupled analysis thrust and torque coefficients for the annular pipe and full contracted geometry. Experimental results are also shown. . 57
5-9 Comparison of experimental and numerical velocity profiles 37% chord axially upstream of IGV ........................................ 58
5-10 Comparison of experimental and numerical velocity profiles 49.7% chord axially downstream of IGV .............................. 59
5-11 Comparison of experimental and numerical velocity profiles 32.2% chord axially downstream of rotor trailing edge .................. 60
5-12 HIREP Rotor Tip-gap Velocity. Note the velocity vectors indicate the flow from pressure to suction side, which results in "roll-up" of the tip-vortex (Duct omitted for clarity) ......................... 61

6-1 Computational grid for RANS modeling of flow domain for Ka4-55 propeller. The main figure illustrates the decomposition of the flow field into several blocks or zones. The upper left inset figure illustrates the discretization over the axial and radial extent of the rotor (not shown). The final inset figure shows leading edge grid details. ...... 64
6-2 Axial velocity contours and axial-radial streamlines for the Ka4-55 duct and shaft geometry. Note that at this stage the RANS code does not take into account the presence of the propulsor (show here merely to indicate prop. location) ................................. 65
6-3 Ka4-55 Blade, hub and duct vortex lattice. The blade is input to PBD as a B-spline control polygon. The hub and duct vortex-lattice surfaces are discretized using the hub and duct streamlines from the RANS flow field. .................................................. 66
Variation of $K_T$ and $K_Q$ with coupling cycles. Each symbol represents an analysis data point using $PBD-14$. Advance coefficient is 0.36.  

Converged RANS flow field following iteration of the coupling process. The presence of the rotor is modeled by body forces in the RANS domain. The presence of the propeller has altered the flow over the duct such that flow now remains attached (Compare with Figure 6-2).  

Comparison of numerical and experimental thrust and torque coefficients for the Ka4-55 propeller in Nozzle 19 over a range of advance coefficients.  

Comparison of radial distribution of circulation for zero and finite tip-gaps.  

Modified MIT Sirenian rotor design incorporating complex tip fin geometry inside a duct!  

HIREP radial pressure probe measurement locations.  

DTRC Propeller 4119.  

Propeller 4119: Comparison of spanwise solved circulation distribution using both methods of hub representation at $J = 0.833$.  

DTRC Propeller 4497.  

Three zone flow domain representation for MIT-PRESWIRL case.  

Convergence of thrust and torque coefficients for both hub models: hub images and vortex lattice hub; with and without blade thickness.  

NSMB Propeller R4-55.  

Comparison of circulation distributions for NSMB R4-55 propeller in uniform inflow; $PBD-14$ versus results by Van Houten. The $PBD$ circulation distribution using the method of images is also shown.
List of Tables

3.1 Comparison of PBD-14 thrust and torque coefficients for varying blade, hub and duct discretizations. The numbers of hub, duct circumferential elements shown represent the discretization over one blade interval. 39

3.2 Comparison of PBD-14 thrust and torque for fixed blade chordwise discretization and varying blade spanwise, hub and duct vortex lattice discretizations. The numbers of hub, duct circumferential elements shown represent the discretization over one blade interval. 40

5.1 HIREP Flow Domain Discretization: Number of grid points 52
5.2 HIREP Rotor Vortex Lattice Discretization 55
5.3 HIREP rotor thrust and torque at $J = 2.31$ 56

6.1 RANS grid discretization of Ka4-55 flow domain 63
6.2 Comparison of experimental and numerical thrust and torque coefficients for the Ka4-55 case 67

B.1 Geometry of DTRC Propeller 4119. 79
B.2 Comparison of DTRC 4119 thrust and torque coefficients at design advance coefficient: $J = 0.833$ 80
B.3 Geometry of NSMB Propeller R4-55. 85
B.4 R4-55 Band Geometry 85
Nomenclature

Mathematical Notation

\( C_{dv} \) \( \frac{Q}{2\rho V_s^2 \pi R^3} \) blade section drag coefficient (twice the frictional drag coefficient)

\( C_Q \) \( \frac{1}{2} \rho V_s^2 \pi R^3 \) torque coefficient based on ship speed

\( C_s \) leading edge suction coefficient

\( C_T \) \( \frac{T}{2\rho V_s^2 \pi R^2} \) thrust coefficient based on ship speed

\( D \) propeller diameter

\( dv \) jump velocity across vortex sheet

\( G \) \( \frac{\Gamma}{2\pi RV_s} \) non-dimensional circulation

\( J_a \) \( \frac{nD}{V_s} \) advance coefficient based on mean inflow velocity

\( J_s \) \( \frac{nD}{V_s} \) advance coefficient based on ship speed

\( K_Q \) \( \frac{Q}{\rho n^2 D^5} \) torque coefficient based on rpm

\( K_T \) \( \frac{T}{\rho n^2 D^4} \) thrust coefficient based on rpm

\( n \) propeller revolutions per second

\( n \) normal vector on a surface

\( P \) pressure

\( Q \) propeller torque

\( R \) propeller radius

\( \mathfrak{R} \) \( \frac{u}{\nu} \) Reynolds number

\( r_h \) hub radius

\( r_i \) vortex image radius

\( r_v \) vortex radius

\( r/R \) local radius

\( s \) curvilinear distance along camber surface

\( t \) unit vector tangent to leading edge

\( T \) propeller thrust
\( u \) velocity
\( \mathbf{V} \) velocity
\( V_a \quad V_s(1 - w) \) velocity of advance
\( V_s \) ship speed
\( \mathbf{V}^\circ \) potential flow velocity
\( \mathbf{V}_c^\circ \) potential flow effective velocity
\( \mathbf{V}_i^\circ \) potential flow circumferential mean induced velocity
\( \mathbf{V}_e^\circ \) potential flow circumferential mean effective velocity
\( \mathbf{V}_e^\circ \) potential flow non-circumferential mean induced velocity
\( \mathbf{V}^\circ \) velocity in axisymmetric flow code
\( \mathbf{V}_e^\circ \) effective velocity in axisymmetric flow code
\( \mathbf{V}^\circ \) circumferential mean velocity in axisymmetric flow code
\( \mathbf{V}_e^\circ \) effective circumferential mean wake in axisymmetric flow code
\( \mathbf{V}^\circ \) non-circumferential mean velocity in axisymmetric flow code

\( w \) Taylor wake fraction
\( \mathcal{Z} \) Number of propeller blades
\( \Gamma \) dimensional circulation
\( \gamma \) vorticity
\( \gamma_b \) bound vorticity
\( \gamma_f \) free vorticity
\( \mu \) vorticity
\( \nu \) kinematic viscosity of fluid
\( \rho \) density of fluid

**Abbreviations**

horseshoe a vortex structure composed of: a spanwise vortex, its shed chordwise vortices on the blade and in the transition wake, and the ultimate wake helices.

CMF circumferential mean force
CMV circumferential mean velocity
LBF local blade force
LBV local blade velocity
LDV Laser Doppler Velocimetry
RANS Reynolds-averaged Navier-Stokes (flow solver)
RMV RANS solution CMV normal to the blade
VEF effective inflow normal to blade
Chapter 1

Introduction

1.1 Overview

Current underwater vehicle design trends include the possible use of single- and multi-stage ducted propulsors combined with a highly tapered after-body [1]. The added complexity of such propulsors requires the use of sophisticated numerical tools for their design and analysis, often analyzing the propulsor in the presence of the body. Such numerical design and analysis techniques [2] include:

1. Lifting-Line Methods
2. Vortex-Lattice Lifting-Surface Methods
3. Boundary Integral Equation Methods
4. Axisymmetric and 3-D Reynolds Averaged Navier Stokes Techniques
5. Streamline Curvature Methods (primarily for turbomachinery and pump-jet applications)

Obviously, all of the above numerical tools have advantages and disadvantages, and advocates of various methods will argue strongly in their favor! However, the Vortex-Lattice Lifting-Surface technique provides a robust, accurate and computationally efficient tool for the design and analysis of complex marine propulsors. Such a lifting surface technique was developed by Kerwin and Lee [3] and has been used extensively, and greatly enhanced [4][5], since its introduction.

Ducted propulsors, where the propeller is surrounded by a shroud or nozzle, offer some advantages over "open" propellers:

- Increased ideal efficiency
- Retardation of Propeller Cavitation
Figure 1-1: Ducted Propulsor - the duct enclosing the propeller is often referred to as a shroud or nozzle.

- Wider operating "window" in terms of efficiency versus advance coefficient
- Provides protection for propeller blades

Two type of duct design are generally used: (1) *Accelerating* duct (Kort nozzle) whereby the duct increases the flow over the propeller and itself produces a net thrust and (2) *Decelerating* duct - the duct produces a net drag force, however, it serves to reduce the flow into the propeller [6]. The first type of duct is used for heavily loaded propeller applications (tug-boats / tankers etc.) where the duct permits a more uniform load distribution over the rotor blade and increased "load-carrying" capacity of the blade at the tip. The latter type of duct is used more in applications where reduction in propeller cavitation noise is desirable. Specifically, for tactical reasons, the reduction of propulsor noise is an important consideration in the design of naval submarine propellers [6].

However, the duct has the disadvantage of increasing the viscous drag on the propulsion unit. In addition, the use of a duct increases the structural complexity of the propulsor.
The Vortex-Lattice Lifting-Surface technique is a potential flow method, and as such has some shortcomings as compared to full viscous models. However, in recent years a coupling procedure has been developed by Kerwin et al. [4], whereby the potential flow lifting surface code is used in conjunction with a Reynolds Averaged Navier Stokes ( RANS ), viscous flow solver. This coupling procedure facilitates analysis of multiple blade rows and reduces computation time ( compared to full three-dimensional viscous calculations ) while still providing an accurate model of the physical behavior.

The potential flow representation does not, however, accurately model the flow through the tip clearance region ( between blade tip and duct inner surface - see Figure 1-2 ). The resulting potential flow solution under-predicts the thrust and torque on the rotor. This deficiency in the current vortex-lattice method has motivated the implementation of a semi-empirical, numerical tip-gap flow model in an effort to better approximate the complex flow patterns in this region.

As an initial step in the implementation of a tip-gap flow model in the existing vortex lattice code ( hereafter designated PBD-14 ), a revised hub and duct model was
incorporated into the program. Previously, the presence of the rigid boundaries of the hub and duct in potential flow was accounted for using the Method of Images. This technique implemented the Kinematic Boundary Condition in an approximate sense i.e. there still existed a component of velocity normal to the surface. An improved hub and duct representation was implemented using a vortex lattice representation of the hub and duct surfaces, on which the Kinematic Boundary condition (zero flux through the rigid boundary), was imposed directly.

In summary, the primary objectives of this work can be stated as follows:

- Representation of the hub and duct as vortex-lattice surfaces
- Implementation of a numerical tip-gap flow model
- Validation of the coupled propeller design and analysis method, including the tip-gap, using experimental data.

In addition, this thesis aims to provide the non-specialist with an insight into aspects of numerical modeling of the complex flow patterns associated with modern marine vehicle propulsors.

1.2 Background

1.2.1 Marine Propulsor Design and Analysis Tools

A comprehensive overview of the vortex lattice method as applied to marine propulsors is provided by Kerwin and Lee [3]. As vortex-lattice lifting-surface methods evolved, their use was expanded to deal with more complex propulsor geometries, for example: ducted propulsors [7] and banded propellers [8].

In PBD-14 a vortex lattice geometry is used to represent the blade mean camber surface. The program facilitates design and analysis of single- and multi-stage propulsors and can be used with a specified inflow velocity field or in conjunction with a Reynolds Averaged Navier Stokes (RANS) solver. Recent versions have represented the blade mean camber surface as a uniform cubic B-spline surface. Blade thickness is represented using a potential flow source distribution over the blade chord.

1.2.2 Hub and Duct Modeling

Previous attempts at modeling the presence of the hub and duct in the vortex-lattice code concentrated on using the Method of Images. However, Van Houten [8] used a vortex lattice to represent a propeller band or duct over the axial extent of the blade.
Brown et al.[7] describe the method of using vortex rings to model the presence of the duct.

Wang [9] represented the hub as a vortex-lattice surface in a previous version of PBD, however, this was not extended to model the duct.

1.2.3 Tip-gap Flow Modeling

The realization that the tip-gap flow constitutes a serious deficiency in the modelling of propulsors using potential theory has motivated numerous authors to address this issue and include some account of the tip-leakage flow in the inviscid model. In the case of both Vortex-Lattice Lifting-Surface and Boundary Integral Equation Methods, one of two approaches has been taken to approximating the viscous flow in the gap using a modified potential flow technique.

"Reduced Equivalent Gap" Method In this method the viscous flow in the tip-gap is modeled by the equivalent inviscid flow through a tip-gap of reduced radial dimension.

Porous Foil An alternative technique of modelling the flow through the tip clearance is to extend the blade to meet the duct and to specify a modified kinematic boundary condition of this portion of the blade. In effect, the boundary condition here allows flow through the blade, the extent of which is governed by a semi-empirical constant. This is known as the "porous foil" technique, developed by Kerwin [10].

1.3 Validation Cases

Validation studies of the expanded coupled viscous propeller blade design and analysis technique were performed using data from large scale experiments on a two stage turbomachine, the HIREP (HIgh REynolds Number Pump Facility) case [11] [12]. This test case also provided a source for validation of the tip-gap model as this was essentially a ducted rotor-stator combination. In addition, the Netherlands Ship Model Basin (NSMB) data for open-water tests on a B-series ducted propeller was also used as a source of validation.

In each case a brief description of the relevant experimental data is provided, followed by an outline of the coupled analysis and finally comparison of experimental and numerical results.
Chapter 2

Coupled Lifting-Surface Design / Analysis Methodology

2.1 Overview

The analytical shortcomings of potential-based propulsor design and analysis methods, in terms of the inviscid analysis, are offset by coupling a viscous flow solver with the vortex lattice method. Such a technique was developed by Kerwin et al. [4] [13] and uses an axisymmetric, Reynolds Averaged Navier Stokes (RANS) based flow solver (designated DTNS). This procedure was originally applied to the design of marine propulsors but has since been extended to include propeller analysis [14]. The primary advantages of using such a method are as follows:

1. Increased computational efficiency over full 3D or axisymmetric RANS codes modeling blades and body.

2. Supports multiple-blade row propulsors. In pure potential methods multi-stage propulsors introduce added complexity as the trailing wake from the upstream stage may interfere with the downstream singularities.

This section provides a descriptive and theoretical outline of the RANS - potential flow propeller design / analysis coupling technique.

2.2 Vortex Lattice Method

The vortex-lattice method can be summarized in three steps:

1. Discretize the geometry

2. Determine Influence Coefficients and impose the Kinematic Boundary Condition

3. Solve the linear system of equations
For lifting surfaces, either hydrofoils or propeller blades, circulation and thickness distributions can be discretized using a lattice or mesh of chordwise and spanwise straight line vortex and source elements respectively. The vortex and source elements are therefore colocated, with their endpoints on the blade mean camber surface. Greeley and Kerwin [15] describe various vortex-lattice spacing options, for example, uniform or cosine spacing. Since Kelvin's Theorem dictates that vortex lines cannot end in the flow field, these vortex elements also have trailing segments, which, together with the bound element, form a horse-shoe vortex [15].

Given that the blade can be represented in this manner, the next step is to implement the appropriate boundary condition on the rigid body. The Kinematic Boundary Condition (KBC) is such that there is no flow through the surface i.e. \( \mathbf{V} \cdot \mathbf{n} = 0 \) - the component of total velocity normal to the blade surface is zero.\(^1\) The KBC is specified at a number of control points, equal to the total number of vortex elements representing the blade.

The vortex-lattice technique used as the basis of this work, \textit{PBD-14}, uses a uniform cubic B-spline representation of the blade mean-camber surface [4]. This compact representation facilitates rapid interrogation of the B-spline control polygon to discretize the blade. Blade thickness is modeled using source elements colocated with the vortex elements. Previous versions of \textit{PBD} used the Method of Images to represent the hub and duct. The current formulation uses a generalized stream-tube geometry to represent the propeller inflow field.

2.2.1 Propulsor Design and Analysis

Propulsor Design Propeller design using the vortex lattice technique involves an iterative approach. In this case an initial trial blade geometry and known flow field are specified. The method then involves manipulating the blade shape in order to enforce the kinematic boundary condition on the blade mean camber surface i.e. \( \mathbf{V} \cdot \mathbf{n} = 0 \). The radial circulation distribution on the blade is known \textit{a priori} and therefore the blade is manipulated to null the normal component of velocity at each blade control point. The axial, radial and tangential velocity induced by each horseshoe vortex on each control point is expressed as an Influence Coefficient matrix, based on the induced velocity due to a horseshoe of unit strength.

\(^1\)For the present, we will consider the velocity here to be the total velocity. A more rigorous description of propeller inflow is provided in subsequent sections.
**Propulsor Analysis** The analysis problem requires the determination of the individual horseshoe vortex strengths, and hence blade circulation, given a particular blade geometry and inflow field. Again, the horseshoe influence coefficients are determined for the given blade geometry and the horseshoe strengths are then determined by direct solution of the matrix equation (Equation 2.1).

For the purpose of the present work the coupling procedure was used purely as an analysis tool. The vortex-lattice representation of the hub and duct is currently only applicable to propulsor analysis. In order to use this enhanced hub / duct representation for propulsor design, the circulation distribution on these surfaces must be known in advance. This is possible if an appropriate duct circulation can be specified at the outset, otherwise a more complex iterative approach to propeller design is required (See Recommendations, Section 7.3).

### 2.2.2 Solution Procedure

Whether used in design or analysis mode, the vortex lattice formulation can be expressed as follows:

\[
[HIF_{ij}][\Gamma_j] = -[V \cdot \hat{n}]
\]  

(2.1)

where

- \([HIF_{ij}]\) is the matrix of Horseshoe Influence Coefficients. The subscripts denote the influence of the \(j^{th}\) horseshoe vortex (vortex element and associated trailing vortices) on the \(i^{th}\) control point. Each element represents the component of velocity **normal** to the surface, induced at a control point due to a horseshoe vortex of unit strength. At this point elements of this matrix can be considered **generalized** influence coefficients. Depending on the inflow specified, these coefficients may take different forms, as explained in Section 2.5.2.

- \([\Gamma_j]\) is the blade circulation vector composed of the individual vortex strengths of the \(j^{th}\) vortex.

- \(V \cdot \hat{n}\) is the component of velocity normal to the blade mean camber surface at the \(i^{th}\) control point.

### 2.3 Propulsor Inflow

A critical aspect of modeling the propulsor is the accurate representation of the propeller inflow. This section provides the relevant background fundamental to the later
discussion of the Coupled Lifting-Surface Design / Analysis Methodology (Section 2.5.1 below). Three important concepts for propeller inflow are:

1. **Nominal Inflow**: Velocity field at the plane of the propeller when there is no propeller operating.

2. **Total Inflow**: Velocity field at the propeller plane when the propeller is operating.

3. **Effective Inflow**: Total inflow less propeller induced velocities.

When using a viscous flow solver in conjunction with the potential flow propeller design / analysis technique the problem can be viewed in terms of two distinct flow regimes:

**Hull Problem** The flow around axisymmetric bodies is analyzed using the RANS viscous flow solver. Using the notation of existing literature on this subject, velocity terms pertaining to the hull problem are denoted as follows:

\[ \mathbf{V}^\circ \]  \hspace{1cm} (2.2)

**Propeller Problem** This aspect of the coupled analysis technique deals specifically with the flow resulting from the operation of a propeller in an specified axisymmetric **effective** inflow. Flow parameters relating to the propeller, or blade, are denoted by the superscript shown below.

\[ \mathbf{V}^\circ \]  \hspace{1cm} (2.3)

The total velocity in the blade problem can therefore be decomposed as follows:

\[ \mathbf{V}^\circ = \mathbf{V}_c^\circ + \mathbf{V}_i^\circ + \mathbf{V}_{i'}^\circ \]  \hspace{1cm} (2.4)

The terms on the right hand side of Equation 2.4 denote the effective inflow (\( \mathbf{V}_c^\circ \)), the circumferential mean induced velocity (\( \mathbf{V}_i^\circ \)) and a fluctuating component of the propeller induced velocity (\( \mathbf{V}_{i'}^\circ \)). Therefore, the total velocity is composed of the effective inflow plus the net propeller induced velocity.

As the coupling procedure uses an axisymmetric viscous flow solver, it is assumed that the effective inflow is also axisymmetric [4]. This facilitates extraction of the appropriate velocity field from the converged viscous flow solution for use in the potential flow propeller code. Kerwin et al.[4][5] provide a more comprehensive description of the velocity decomposition.
2.4 **Viscous Flow Solver**

The axisymmetric viscous flow solver used in the coupling procedure is based on a Reynolds Averaged Navier Stokes Solver (RANS) computation technique. The flow domain is represented as a two dimensional grid over which the RANS equations are solved using a finite-volume technique. Valentine [16] provides a comprehensive overview of the application of RANS codes to the field of Marine Propulsors. Specifically, Valentine points out that in the design of vehicles with a full stern hull form, the viscous flow solver plays an important role. Conventional hull forms are optimized for low total resistance, resulting in streamlined hull geometries. When a highly tapered after-body is considered, the flow will generally separate off the stern in the absence of the propulsor. The design of the propulsor as an integrated part of the hull concept therefore requires investigation of the ability of the propulsor to prevent flow separation. The viscous nature of this phenomenon suggests the use of a RANS flow solver.

The RANS flow solver currently being used in conjunction with the potential flow propeller design and analysis code was developed by Gorski [17]. This axisymmetric formulation solves the Reynolds Averaged Navier Stokes equations using a finite volume approach. The method therefore uses a structured grid, but facilitates use of multiple four-sided "zones", subject to the constraint that only a single boundary condition can be applied to each zone boundary. In addition, either the Baldwin-Lomax or $k - \varepsilon$ turbulence models [16] can be used when modeling turbulent flow.

2.5 **Coupled Lifting Surface Design / Analysis Methodology**

Coupling the potential flow propeller blade design / analysis technique with a RANS flow solver involves an iterative process whereby the two distinct methods are linked at a common interface. This interface facilitates "cross-talk" between the different methods. The interaction is shown in Figure 2-1. The iterative coupling procedure revolves about a "hub" involving the extraction of a flow field from the solution of the viscous flow around the body and the transfer of propeller forces from the propeller analysis code to the flow solver.

This section provides a descriptive overview of the coupling procedure prior to outlining the mathematical basis for the formulation.
2.5.1 Descriptive Overview

The coupling procedure follows the path outlined in Figures 2-2 and 2-3. Following discretization of the flow domain as a two-dimensional computational grid, the RANS flow solver is invoked to obtain a converged flow solution over the body in the absence of the propulsor. A velocity field is extracted from the converged RANS flow solution in the region of the propeller plane for input to the potential flow domain. Using this inflow, the vortex-lattice method is used to analyze the propeller.

Propeller thrust and torque coefficients are obtained from the vortex lattice analysis of the propeller. The blade forces are then overlaid on the computational grid, appearing as body forces in the Reynolds Averaged Navier Stokes equations. Since an axisymmetric flow solver is being used these forces correspond to the Circumferential Mean (CMF) forces [4].

The coupling procedure now follows the circular path shown in Figure 2-3. The RANS flow solver is rerun with the additional body forces, now representing the presence of the propeller. It should be noted that multiple blade rows can easily be accommodated using this technique. The forces from each propulsor stage are
"superimposed" on the RANS grid. Upon convergence of the flow solver, the velocity field in the region of the propulsor is again extracted and used for the subsequent propeller analysis using the vortex-lattice technique.

This iterative process is pursued until the rotor thrust and torque coefficients converge. The procedure is robust in that convergence is generally arrived at following eight to ten cycles between the various components\(^2\).

![Flowchart of steps involved in RANS - potential flow propeller design / analysis coupling](image)

Figure 2.2: Flowchart of steps involved in RANS - potential flow propeller design / analysis coupling

### 2.5.2 Mathematical Formulation

The preceding section served to provide an overview of the coupling procedure. It now remains to present the coupled analysis problem in a more rigorous format.

A generalized solution procedure was presented in Section 2.2.2. Recall:

\[ [\text{HIF}_{ij}] [\Gamma_j] = -[V \cdot \hat{n}] \]  

(2.5)

In this equation \([\text{HIF}_{ij}]\) denotes a generic influence coefficient, which may depend on the specified inflow. This notation can now be expanded to describe two types of influence coefficients:

---

\(^2\)This is merely a guideline as the number of iterations will vary from case to case. Also, the number of RANS flow iterations will vary depending on the complexity of the geometry / Reynolds Number etc.
Local Blade Velocity influenced coefficients, denoted $[\text{LBV}_{ij}]$, represent the total velocity induced at the $i^{th}$ control point by the $j^{th}$ vortex lattice element.

Circumferential Mean Induced Velocity influence coefficients representing the circumferential mean induced velocity at the $i^{th}$ control point by the $j^{th}$ vortex lattice element. Denoted by $[\text{CMV}_{ij}]$.

While these influence coefficients are calculated as three-component functions, when solving the matrix system the normal influence coefficients are used, such that they are consistent with the normal velocity on the right hand side of Equation 2.5.

Equation 2.5 can be rewritten as:

$$[\text{LBV}_{ij}] [\Gamma_j] = - [\text{VEF}_i] \quad (2.6)$$

The term on the right hand side denotes the propeller effective inflow. When performing a coupled analysis using an axisymmetric solver, Equation 2.6 highlights a problem! The propeller inflow extracted from RANS is obtained using circumferential mean body forces from the previous propeller analysis iteration. The effective inflow should not include these velocity components and therefore they must be extracted from the inflow field. Ideally, this effective wake problem could be formulated
as follows (using the notation of Kerwin et al. [5]):

\[ [LBV_{ij} - CMV_{ij}] [\Gamma_j] = -[VEF_i + RMV_i] \] (2.7)

with

\[ [RMV_{ij}] = [CMV_{ij}] [\Gamma_j]^{n-1} \] (2.8)

where

- \([RMV_{ij}]\) denotes the component of velocity in the RANS solution due to propeller induction.

- \([\Gamma_j]^{n-1}\) denotes the circulation distribution obtained from the previous propeller code iteration.

To expand on this point, the ideal formulation would be to solve for the vortex strengths using an influence coefficient matrix which already has the circumferential mean induced velocity component subtracted away! This would then be equivalent to the RANS inflow with the CMV propeller induced velocities deducted - essentially, the effective wake. This is the optimal method by which to solve the system. However, the component of the RANS inflow due to the propeller CMV velocities can only be determined using values obtained from the previous vortex lattice analysis, for reasons outlined later in this section.

Returning to the problem in hand: The analysis problem requires the determination of the individual vortex strengths, given a prescribed propeller geometry and solved inflow. Thus, Equation 2.7 can be rearranged in terms of the desired circulation distribution.

\[ [\Gamma_j] = -[LBV_{ij} - CMV_{ij}]^{-1} [VEF_i + RMV_i] \] (2.9)

During initial formulation by Kerwin et al. [4], it was found that the solution procedure outlined in Equation 2.9 was unstable, possible due to the ill-conditioned nature of the influence coefficient matrix. A scheme was therefore adopted whereby the circumferential mean induced velocity influence coefficient matrix was moved to the right hand side of the above equation.

\[ [LBV_{ij}] [\Gamma_j] = -([VEF_i + RMV_i] - [CMV_{ij}] [\Gamma_j]) \] (2.10)

However, the unknown variable, \([\Gamma_j]\) now appears on both sides of the equation. Since the coupling procedure is an iterative scheme, the problem can be reformulated as follows:

\[ [LBV_{ij}] [\Gamma_j] = -([VEF_i + RMV_i]^{n-1} - [CMV_{ij}] [\Gamma_j]^{n-1}) \] (2.11)
where $[\Gamma_j]^{n-1}$ represents the solved circulation distribution from the previous execution of the vortex-lattice technique.

$$[\Gamma_j] = -[LBV_{ij}]^{-1}([VEF_i + RMV_i^{n-1}] - [CMV_{ij}][\Gamma_j]^{n-1})$$  \hspace{1cm} (2.12)

Using a simplified notation:

$$[\Gamma_j] = -[LBV_{ij}]^{-1}(\text{INFLOW}_{RANS} - [\text{PREVIOUS ITERATION'S CMV}_i])$$  \hspace{1cm} (2.13)

Equation 2.13 is really an oversimplification of the problem! The effective inflow on the right hand side will have components due to the propeller rotation and the source induced velocities, in addition to the \textit{RANS} inflow.
Chapter 3

Potential Flow Modeling of Hub and Duct

3.1 Overview

Potential flow theory is a robust, efficient method of analyzing hydrodynamic systems. In the case of marine propellers, potential flow methods (in the form of vortex-lattice techniques) have been used successfully in both design and analysis. In modeling the propulsor using a vortex-lattice lifting surface, it is desirable to accurately model the presence of the hub and duct as this will influence the circulation distribution over the blade span, and consequently the blade forces. In the absence of a hub or duct, Kelvin’s Theorem dictates that the circulation must be zero at the extremities of the blade. If this was not the case the fluid layer outboard of the tip would have to support a finite pressure jump, resulting in infinite shear stresses within the fluid [18]. Conversely, the presence of a hub, duct or both will enable the blade section sustain some specified load at its outer extremities.

Previously, the Method of Images was used in PBD to represent the hub and duct. This was found to accurately model the presence of the hub, however, some discrepancies were evident when using images to model the presence of a duct. Also, the vortex element images were satisfactory from the point of view of imposing the KBC, however, when imaging the blade thickness source elements, the method of images was less accurate. Therefore, a new, more rigorous method of modelling the hub and duct was formulated whereby the hub and duct are represented as vortex-lattice surfaces.
3.2 Method of Images

The Method of Images provides a means of implementing various boundary conditions in potential flow theory. In the case of a singularity near a rigid boundary, the boundary obviously represents a streamline of the flow-field. This is represented in Potential flow by introducing an additional singularity into the flow field at such a position as to mirror the original singularity.

The Method of Images is best illustrated by a simple 2-D example. Consider a point vortex in an infinite fluid. In order to represent a solid hub a point vortex "image" is placed inside the desired physical hub radius. The radial dimension of the image is determined as follows:

\[ r_i = \frac{r_h^2}{r_v} \]  

(3.1)

Where the subscripts denote the radius of the image vortex, hub and actual vortex. This is illustrated in Figure 3-1, the 2D point vortex being located at [0, 1]. By placing an image vortex inside the desired hub boundary, it is observed that the hub radius now forms a streamline of the flow.

![Streamlines Illustrated](image)

Figure 3-1: 2D Hub Image. Placement of point vortex and image create streamlines at radius = 0.5

The simple 2-D point vortex example shown in Figure 3-1 can be extended to
encompass vortex and source element imaging in three dimensions. The resulting image lattice is shown in Figure 3.2. Note that the image singularities are positioned using the blade pitch of the tip (for the duct) or root (for the hub).

![Figure 3-2: PBD Hub and Duct Images](image)

### 3.3 Vortex Lattice Hub / Duct

An improved method of modeling the Hub and Duct in PBD-14 was devised whereby the hub and duct are represented by vortex-lattice surfaces on which the kinematic boundary condition, essentially zero flux through the rigid boundary, was imposed directly. This is a similar approach to that used in Boundary Integral Methods applied to propulsor design, for example, Kerwin, Kinnas et al. [19].

#### 3.3.1 Hub and Duct Geometry

Beginning with the existing blade lattice discretization the hub and duct vortex lattice geometry was determined as follows:

**Blade Region** Discretization of the hub and duct over the axial extent of the blade was constrained by the chord-wise vortex lattice spacing algorithm used on the
blades (either uniform or cosine spacing). The axial positions of the hub and duct elements were constrained to be coincident with these points. A blended spacing algorithm (described below) was used for the circumferential discretization of the hub and duct. The axial positions of the control points were obtained by algebraic manipulation of the first span-wise blade vortex lattice elements. The Y-Z co-ordinates were determined by cylindrical co-ordinate interpolation of the surrounding four vortex lattice grid points (see Figure 3-3).

![Blade Vortex Lattice](image)

**Figure 3-3: Hub vortex lattice in blade region**

**Upstream Discretization** The upstream extent of the hub and duct vortex lattice can be varied according to the physical dimensions of these surfaces. The axial and radial co-ordinates of the upstream vortex lattice are determined by spline-fitting the appropriate streamlines from the input flow-field and subdividing the upstream axial distance using a half-Cosine spacing routine. The circumferential discretization is then carried out using an upstream helical pattern, the pitch angle of which is determined by the respective blade root or tip pitch angle.

**Downstream Discretization** The vortex lattice downstream of the blade is governed by the blade transition wake sheet location. If this were not the case a
situation may arise whereby the hub control points were positioned along a blade transition wake element. In addition, the downstream extent of the lattice on the hub and duct is specified as a given number of blade transition wake elements.

Figure 3-4: Vortex Lattice Representation of Hub and Duct. The propulsor shown is the MIT Sirenian case.

3.3.2 Transition Wake Geometry

Using vortex-lattice theory, a surface is represented by discrete "horseshoe" vortex elements. Each discrete vortex element consists of a spanwise vortex element and trailing vortex elements extending chord-wise over the surface, into the transition wake and terminating in an ultimate wake. The blade transition wake is aligned with the local inflow. The hub and duct must also have trailing wake sheets which follow the appropriate streamlines representing the rigid boundaries of these surfaces (Figure 3-5).

3.4 Hub / Duct Circumferential Spacing

Initially, the circumferential spacing of vortex lattice elements on the hub and duct surfaces was implemented using a uniform spacing routine; essentially the blade
interval was subdivided into a preset number of meridional elements. However, in situations where cosine spacing was used in a spanwise direction on the blade, the relative dimensions of the vortex elements at the blade-hub and blade-duct intersections, were disproportionate. This resulted in local aberrations in the circulation distribution, especially at the blade tip.

In order to circumvent this problem a "blended" circumferential spacing algorithm was devised. By this method, the angular spacing interval on the hub and duct was constrained by the following factors:

1. Blade angular interval
2. Dimension of blade vortex element closest to the hub or duct
3. Number of vortex elements constituting the blade interval

The objective was to determine a common ratio between successive angular increments. Representing the sum of all the angular spacings in a half-blade interval as the sum of \( n \) terms in a geometric series:

\[
S_n = \frac{a(1 - r^n)}{(1 - r)}
\]  

(3.2)
where:

- $S_n$ is the partial sum of the geometric series (of $n$ terms)
- $a$ defines the first term in the series
- $r$ is the common ratio between successive terms

For this application, $S_n$ denotes the half-blade interval, $a$ defines the first angular increment closest to the blade on the hub or duct and $n$ is the number of circumferential spacings on the hub / duct over the half-blade interval. The parameter $r$, the common ratio, must be determined.

Reformulating Equation 3.2 above:

$$\theta_{b/2} = \frac{\theta_1(1 - r^n)}{(1 - r)} \quad (3.3)$$

where:

- $\theta_{b/2}$ is the half-blade angular increment $= \frac{2\pi}{Z}$ where $Z$ is blades.
- $\theta_1$ defines the first angular interval

Manipulation of the above expression (Equation 3.3) yields an equation from which the common ratio can be determined:

$$\frac{\theta_{b/2}}{\theta_1} = 1 - \frac{r^n}{1 - r} \quad (3.4)$$

$$r^n - \frac{\theta_{b/2}}{\theta_1} r = 1 - \frac{\theta_{b/2}}{\theta_1} \quad (3.5)$$

$$r^n - \frac{\theta_{b/2}}{\theta_1} r + \frac{\theta_{b/2}}{\theta_1} - 1 = 0 \quad (3.6)$$

A Newton-Rhapson iterative scheme was used to solve for $r$ in Equation 3.6.

$$r_{i+1} = r_i - \frac{f(r)}{f'(r)} \quad (3.7)$$

Inspection of Equation 3.6 reveals the fact that $r = 1.0$ will always be a solution. This is undesirable and a second root is required. This problem is resolved by careful selection of the initial guess for the iterative scheme, $r_0$. The following criteria appears to identify the desired root under all conditions:

$$if \quad \frac{\theta_{b/2}}{\theta_1} > n \implies r_0 = 10.0 \quad (3.8)$$

$$if \quad \frac{\theta_{b/2}}{\theta_1} < n \implies r_0 = 0.0 \quad (3.9)$$

(3.10)
3.5 Computational Aspects of Hub, Duct Modeling

3.5.1 Method of Images

The general solution procedure for propulsor design and analysis is illustrated in Section 2.2.2. In determining the Influence Coefficients, the effect of a given vortex element also includes the influence of its images. This is true for both vortex and source elements. The influence coefficient matrix is then composed solely of the effect of all the blade vortex elements on the blade control points, essentially a "blade-on-blade" problem.

3.5.2 Vortex Lattice Hub and Duct

With the extension of the system to include a vortex lattice hub and duct, the propulsor model can now be defined in terms of three distinct surfaces.

1. The original blade vortex lattice / control points
2. Hub vortex lattice and associated control points
3. Duct vortex lattice and control points

Formulation of the overall Horseshoe Influence Coefficient matrix is now more complicated as the influence of each surface on the three sets of control points must be considered. The resulting matrix equation is:

\[
\begin{bmatrix}
\text{Vortex Elements} \\
\text{Blade} & \text{Hub} & \text{Duct} \\
\hline
\text{Blade} & \text{Control Points} & \text{Hub} & \text{Duct} \\
\text{Blade} & \text{Hub} & \text{Duct} \\
H_{\text{BB}} & H_{\text{HB}} & H_{\text{DB}} \\
H_{\text{BH}} & H_{\text{HH}} & H_{\text{DH}} \\
H_{\text{BD}} & H_{\text{HD}} & H_{\text{DD}}
\end{bmatrix}
\begin{bmatrix}
\Gamma_j
\end{bmatrix}
= -\left[ \mathbf{V} \cdot \mathbf{n} \right] 
\tag{3.11}
\]

where the superscripts denote the influence of a vortex lattice surface on a set of control points; for example, \( H_{\text{DB}} \) denotes the influence coefficients relating the velocity induced by the Duct vortex lattice on the Blade control points.\(^1\)

This extended system is solved in the same manner as the blade-image problem as described in Section 2.2.2. The right hand side of 3.11 is formulated by calculating the velocity component normal to the surface. Again, the stated influence coefficients are normal velocity influence functions.

\(^1\)Superscripts : B = blade ; H = hub and D = duct
3.6 Verification of Solution Procedure for Vortex Lattice Hub and Duct

The Method of Images imposes an approximate boundary condition on the physical hub surface. In the case of vortex-lattice hub and duct representation the kinematic boundary condition is imposed directly. The resulting normal velocities at the control points using the latter method are zero. This is illustrated in Figure 3-6, the flow being tangent to the blade and hub surface.

![Diagram](image)

Figure 3-6: Total velocity at the Hub and Blade control points. The flow is tangent to the surface, thus illustrating the imposed kinematic boundary condition of zero flux through the surface.

3.7 Role of Hub and Duct in Coupled Analysis

The coupled analysis procedure has been outlined in Section 2.5.2. Adding a hub and duct to the matrix system greatly expands the complexity of the problem. Equation 3.11 above presents the expanded matrix system composed of generalized influence coefficients. For the purposes of performing a coupled analysis this matrix system

36
must be reformatted as follows:

\[
\begin{bmatrix}
\text{Blade} & \text{Hub} & \text{Duct} \\
\text{Blade} & \begin{bmatrix} LBV_{ij}^{BB} & LBV_{ij}^{HB} & LBV_{ij}^{DB} \end{bmatrix} \\
\text{Hub} & \begin{bmatrix} LBV_{ij}^{BH} & LBV_{ij}^{HH} & LBV_{ij}^{DH} \end{bmatrix} \\
\text{Duct} & \begin{bmatrix} LBV_{ij}^{BD} & LBV_{ij}^{HD} & LBV_{ij}^{DD} \end{bmatrix}
\end{bmatrix} [\Gamma_j] = - \begin{bmatrix} VEF_j^B \\
VEF_j^H \\
VEF_j^D \end{bmatrix} \quad (3.12)
\]

Again, as the blade circumferential mean induced velocities are included in the RANS inflow, the matrix system can be expanded to a linear set of equations similar to Equation 2.11:

\[
\begin{bmatrix}
LBV_{ij}^{BB} & LBV_{ij}^{HB} & LBV_{ij}^{DB} \\
LBV_{ij}^{BH} & LBV_{ij}^{HH} & LBV_{ij}^{DH} \\
LBV_{ij}^{BD} & LBV_{ij}^{HD} & LBV_{ij}^{DD}
\end{bmatrix} \begin{bmatrix} \Gamma_j^B \\
\Gamma_j^H \\
\Gamma_j^D \end{bmatrix} = \\
- \begin{bmatrix} (VEF_j^B + RMV_j^{B(n-1)}) - ([CMV_{ij}^B][\Gamma_j]^{(n-1)}) \\
(VEF_j^H + RMV_j^{H(n-1)}) - ([CMV_{ij}^H][\Gamma_j]^{(n-1)}) \\
(VEF_j^D + RMV_j^{D(n-1)}) - ([CMV_{ij}^D][\Gamma_j]^{(n-1)}) \end{bmatrix} \quad (3.13)
\]

At first glance the above equation appears quite complex, however, it is merely an extension of the previous "blade-only" formulation. The global influence coefficient matrix now consists of sub-matrices, for example, a component block is \(LBV_{ij}^{BD}\), the influence of the blade vortex lattice on the duct control points.

Since the hub and duct surfaces are streamlines of the flow, the component of inflow velocity normal to the surface is zero. The blade, hub and duct vortex lattice systems induce zero circumferential mean velocity normal to the hub and duct surfaces. This has, in fact, been shown to be the case by using a field-point velocity calculation routine at the hub and duct control points. Therefore, the second and third rows of the vector on the right hand side of Equation 3.13 will be zero.

In addition, in order to simplify the extent of reliance on past iterations it was decided to formulate the global matrix using "mixed" influence coefficients. Let us first examine this modified system before offering an explanation.
\[
- \left[ \left( \text{VEF}^B + \text{RMV}^B(n-1) \right) - \left( [\text{CMV}^i_{Blade \ on \ blade}] [\Gamma_j]\right)^{(n-1)} \right]
\]

\[
(\text{VEF}^H + \text{RMV}^H(n-1))
\]

\[
(\text{VEF}^D + \text{RMV}^D(n-1))
\]

(3.14)

Returning to the "blade-only" formulation presented in Section 2.5.2, it was stated that the initial effective wake solution procedure, using influence coefficients with the CMV component already deducted (LBV_{ij} - CMV_{ij}), was inherently unstable. This was also found to be the case when the matrix system was extended to include the hub and duct vortex lattices. However, the partial use of mixed influence coefficients in the global matrix did result in a stable solution procedure. Equation 3.14 illustrates the use of this combination of both local blade velocity and circumferential mean induced velocity influence coefficients.

The blade-on-blade sub-matrix in Equation 3.14, as in the blade-only formulation, represents the local blade velocity influence coefficients for the blade vortex lattice on blade control points. Therefore, to solve the effective wake problem, the CMV induced velocity, due to the effect of the blade vortex lattice on blade control points only, must be deducted from the right hand side. Hence the entry in the first row of the velocity vector on the right hand side.

It was found that the CMV blade induced velocity due to the hub and duct lattices could be deducted from the influence coefficients on the left hand side. This eliminated the need to account for these velocities on the right hand side and thus the reliance of the scheme on previous iterations was reduced. The boundary condition on the hub and duct surfaces dictates that the normal component of the total velocity at these control points must be zero. This justifies the use of local blade velocity influence coefficients (LBV_{ij}) on the left hand side.

### 3.8 Convergence Studies

Some initial studies were performed regarding thrust and torque convergence using the extended vortex-lattice representation of the hub and duct. The propeller used as the basis of this investigation was the Ka4-55 ducted propulsor, operating in uniform inflow at an advance coefficient of 0.72. Tip-gap modeling was not applied to this convergence study.

Initially, a series of test runs were performed using a "square" blade vortex lattice i.e. equivalent chordwise and spanwise discretizations. For each increment of the blade discretization a range of circumferential hub and duct discretizations were investigated. The results are presented in Table 3.1. It should be noted that the hub
and duct discretizations reported here relate to the number of meridional elements in a single blade interval. This convergence test highlighted some important aspects of grid density selection for the blade, hub and duct.

1. The lowest density of vortex lattice elements on the blade, an $8 \times 8$ lattice, provides results to within 5% of the higher discretizations. Therefore, for a first "cut" at a propulsor analysis, low density lattices may be sufficient.

2. The variation of thrust and torque coefficients with circumferential hub and duct elements reduces as these values approach acceptable limits. An "acceptable" limit has been identified as the number of circumferential elements in the half-blade interval such that the common ratio between successive circumferential elements in the blended spacing scheme is in the range $0.5 \rightarrow 2.0$ (See Section 3.4).

3. The important aspect of choosing an appropriate discretization is to ensure that the blade-hub and blade-duct discretizations are comparable. An extremely fine blade lattice will be inadequate if the hub or duct vortex lattices are coarse!

4. Following on from the previous point, high blade discretizations require large hub and duct systems - the result being a computationally intensive system. Therefore, for most systems a $16 \times 16$ blade grid is adequate with 14 and 18 circumferential elements over a single blade interval on the hub and duct respectively.

<table>
<thead>
<tr>
<th>Blade</th>
<th>Hub, Duct Circumferential Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>6,10</td>
</tr>
<tr>
<td>$8 \times 8$</td>
<td>0.1619</td>
</tr>
<tr>
<td></td>
<td>0.3046</td>
</tr>
<tr>
<td>$12 \times 12$</td>
<td>0.1203</td>
</tr>
<tr>
<td></td>
<td>0.2575</td>
</tr>
<tr>
<td>$16 \times 16$</td>
<td>0.3614</td>
</tr>
<tr>
<td></td>
<td>0.5908</td>
</tr>
<tr>
<td>$24 \times 24$</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>-</td>
</tr>
</tbody>
</table>

Table 3.1: Comparison of $PBD-14$ thrust and torque coefficients for varying blade, hub and duct discretizations. The numbers of hub, duct circumferential elements shown represent the discretization over one blade interval.

Having identified an appropriate starting point, the convergence analysis was advanced further as follows: given the relative dimensions of the chord and span for most propeller applications, it may be sufficient to use fewer elements across the blade
chord. Especially since the spanwise elements have a more direct influence on the hub and duct spacing. Therefore, using more spanwise elements should prove more efficient. This hypothesis was tested by fixing the number of chordwise elements and varying the spanwise, hub and duct circumferential discretizations. The resulting thrust and torque coefficients are presented in Table 3.2.

<table>
<thead>
<tr>
<th>Blade</th>
<th>Hub, Duct Circ. Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chord</td>
<td>Span</td>
</tr>
<tr>
<td>16</td>
<td>16</td>
</tr>
<tr>
<td>16</td>
<td>20</td>
</tr>
<tr>
<td>16</td>
<td>24</td>
</tr>
<tr>
<td>16</td>
<td>28</td>
</tr>
<tr>
<td>16</td>
<td>32</td>
</tr>
</tbody>
</table>

Table 3.2: Comparison of PBD-14 thrust and torque for fixed blade chordwise discretization and varying blade spanwise, hub and duct vortex lattice discretizations. The numbers of hub, duct circumferential elements shown represent the discretization over one blade interval.

Again, the above analysis highlights the relative dimensions of the component lattices as an important consideration in analyzing the propulsor. Higher discretizations on the blade do not necessarily perform better as this requires excessive hub and duct discretization. For example, using a 16 x 28 blade lattice, the increase in hub and duct elements from 22,26 to 26,30 yields less than 0.15% increase in $K_T$ and $K_Q$.

In summary, the relative dimensioning of the blade, hub and duct systems has been identified as a driving factor in the choice of appropriate discretizations.
Chapter 4

Numerical Implementation of the Tip-gap Flow Model

4.1 Overview

Tip-gap flow in ducted propulsors is one area where the potential flow theory provides a poor approximation to the physical flow characteristics. The viscous nature of the flow in the tip clearance region is poorly represented by the exaggerated potential theory solution. It has been shown [10] that the flow in this region has global effects on the blade circulation distribution and hence the propulsor forces. For this reason an accurate method of numerically modelling the tip-gap flow in the inviscid vortex lattice code is imperative.

Numerical tip-gap flow models generally use a semi-empirical approach. The actual viscous flow through the gap is approximated as a reduced inviscid flow, related by an empirical coefficient derived from experimental data. Van Houten [8] used an equivalent inviscid gap technique in implementing the tip gap flow model. Kerwin [10] approached the problem using a porous foil technique whereby the tip-gap is modeled as a spanwise extension of the blade vortex lattice with a modified kinematic boundary condition permitting flow through the blade, hence the "porous foil".

A tip-gap flow model, based on the porous foil approach [10], was incorporated into the analysis modes of PBD-14.

4.2 Viscous Flow in the Tip-Clearance Region

One of the primary objectives in including a duct into a propulsor design is to provide a more uniform load distribution across the blade span. The presence of the duct of a band enables the propeller blade tip to sustain a finite loading. The extent of this loading depends on the gap dimension. Ideally, the gap should be as small as possible,
yet still be sufficient from the point of view of clearance between the components.

The small tip-clearance dimension results in highly viscous flow in the gap. The viscous effects allow the blade tip to sustain a finite pressure jump. The rotor blade loading produces a large pressure jump across the blade which forces fluid from the pressure side, through the tip-gap, to the suction side of the blade. This flow pattern interacts with (1) the duct inner surface boundary layer, (2) flow separation from the blade tip and (3) the propeller inflow, resulting in complex flow patterns. The tip-leakage vortex "rolls up" into a spiral configuration, under the influence of the accelerated flow in the gap.

4.3 Porous Foil Model

The flow in the tip clearance region can be approximated by that through a 2D orifice:

\[ w_R = C_Q \sqrt{\frac{2\Delta p}{\rho}} \]  \hspace{1cm} (4.1)

where \( w_R \) defines the relative flow through the gap; \( C_Q \) is the Orifice co-efficient and \( \Delta p \) is the pressure jump across the tip of the blade. The pressure jump can be related to bound vorticity on the porous section of the foil:

\[ \Delta p = \rho U_\infty \gamma \]  \hspace{1cm} (4.2)

Physically, no bound vorticity can exist in the free fluid, however, by approximating the gap as part of the blade this apparent paradox is valid. Substituting Equation 4.2 in the orifice equation:

\[ w_R = C_Q \sqrt{2U_\infty \gamma} \]  \hspace{1cm} (4.3)

In the case of a 2D foil at some angle of attack, \( \alpha \), adjacent to a rigid boundary, the flow in the tip-gap region is given as follows:

\[ w_{tipgap} = -U_\infty \alpha + C_Q \sqrt{2U_\infty \gamma_{gap}} \]  \hspace{1cm} (4.4)

where \(-U_\infty \alpha\) denotes the normal component of velocity through the gap panels.

This can be further extended to three dimensions. The normal component of velocity is now given as \(-[\vec{V} \cdot \hat{n}]\). The "mainstream" velocity is now the local inflow tangent to the gap panels.

\[ w_{tipgap} = -[\vec{V} \cdot \hat{n}] + C_Q \sqrt{2[\vec{V} \cdot \hat{t} \gamma_{gap}} \]  \hspace{1cm} (4.5)

By further relating the bound vorticity to the velocity jump across the gap vortex lattice: \( \gamma = 2V_d \) (where \( V_d \) denotes the jump velocity).

\[ w_{tipgap} = -[\vec{V} \cdot \hat{n}] + 2C_Q \sqrt{[\vec{V} \cdot \hat{t}]V_d} \]  \hspace{1cm} (4.6)
4.4 Integration of the Tip-gap Flow Model in \textit{PBD-14}

4.4.1 Blade lattice Modification

In \textit{PBD-14}, the blade vortex lattice was extended by one vortex element in the spanwise direction, the radial dimension of the additional elements being equivalent to the tip-gap height. An additional row of control points was also generated (see Figure 4-1).

![Tip-gap Vortex Lattice and Control Points](image)

Figure 4-1: Tip-gap Vortex Lattice and Control Points

4.4.2 Adjustments to Solution Procedure

A brief outline of the general Vortex Lattice solution procedure was provided in Section 2.2.2. In implementing the tip-gap flow model in \textit{PBD-14}, the Horseshoe Influence Coefficients were calculated as before, however, the vortex lattice grid now contained the modifications outlined above, namely a spanwise extension of the blade by a single row of vortex elements.

In the absence of a tip-gap the right hand side of the matrix equation (Equation 2.1), is formulated by evaluation of the dot product of the blade normal and the local inflow at the respective control point. However, in the case of the tip-gap control
points, a modified boundary condition is specified. Essentially, the flow through the gap is given by the orifice equation. In assembling the right hand side of the matrix equation, both boundary conditions must be satisfied!

4.4.3 Iterative Solution of the Flow in the Tip-gap

Close inspection of Equation 4.6 illustrates the iterative nature of the problem. Initially, the bound vorticity in the gap is unknown. The vortex elements are therefore assigned a value of zero during matrix formulation. This initial gap vortex element strength is used to solve the global problem (Analysis Mode). Having determined the vortex strengths constituting the Blade, Hub and Duct vortex lattices, the local velocity at the gap control points is computed and used to determine the actual gap vorticity using the velocity jump across the tip-gap.

The iterative solution, represented schematically in Figure 4-2, can be stated as follows:

\[ [\Gamma]^n = [HIF_{ij}]^{-1} [\vec{V} \cdot \hat{n}]^{n-1} \]  

(4.7)

For the procedure outlined above the Horseshoe influence coefficients need only be calculated once as the position of the propulsor components is unchanged in Analysis mode.

This iterative process was found to converge rapidly when a relaxation factor was introduced into Equation 4.7.

4.5 Previous Tip-gap Treatment in PBD-14

In previous versions of PBD-14, no account was taken of the viscous nature of the tip-gap flow. Therefore, if an actual tip-gap was specified in the input, the resulting circulation distribution at the blade tip would be lower than for the closed gap case, due to the potential flow in the tip-clearance region. Figure 4-3 represents the spanwise distribution of circulation for the HIRED rotor with varying tip-gap dimensions.

4.6 Tip-gap effects!

In verifying the effect of the tip-gap flow on the spanwise distribution of circulation, and hence the blade thrust and torque, the HIRED rotor (illustrated in Figure 4-1) was analyzed with and without the tip-gap flow calculation routine enabled. The resulting circulation distributions are shown in Figure 4-4. When the presence of
the viscous flow through the tip clearance is considered, the circulation distribution exhibits a substantial decrease, which is confined mainly to the blade extremity.
Figure 4-3: Effects of varying tip-gap dimension on Spanwise Distribution of Circulation. The viscous nature of the flow in the clearance region has NOT been considered. (Note: gap dimensions stated as percentage of rotor diameter)

Figure 4-4: Effect of Tip-gap on Spanwise Distribution of Circulation. These results were derived from the Vortex lattice hub and duct implementation of PBD-14 using the tip-gap flow model (Orifice Coefficient: $C_Q = 0.84$)
Chapter 5

Case I: HIREP

5.1 Overview

The HIREP (HIgh REynolds Number Pump) facility test data [11] [12] provided a basis for validation of the improved vortex-lattice propeller design and analysis code. This series of experiments reported pressure measurements, rotor thrust and torque coefficients and Laser Doppler Velocimetry (LDV) data for a large, two-stage (stator-rotor) turbomachine pump. The pump was operated at a Reynolds Number (based on rotor chord) of approximately $3 \times 10^6$. This internal flow problem was modeled using the coupled viscous-potential flow analysis incorporating the vortex lattice hub and duct implementation and the semi-empirical tip-gap flow model. While the experiment initially appeared promising as a test case for the PBD vortex-lattice technique, uncertainty in both the experimental and numerical results undermined the validation.

The HIREP case has been used by other authors as a validation case for Computational Fluid Dynamics (CFD) techniques. Yang [20] used the HIREP case as a basis for his investigation of flow passage through a multiple-blade row turbomachinery pump, using an implicit, high-resolution finite difference RANS technique.

5.2 Experimental Facility

The experimental facility is described by Zierke at al.[21][12]. Essentially, the HIREP facility consists of a recirculating water tunnel with a $48\text{ in.}$ diameter test section. The tunnel liner diameter is $42\text{ in.}$ and the ratio of hub radius to rotor tip radius, $\frac{r_h}{R}$, is 0.5. The experimental data used for validation was obtained from a series of experiments on a two-stage stator-rotor pump (Figure 5-1). The pre-swirl stator stage has 13 blades while the rotor is a 7-bladed unit. More specific details regarding the stator and rotor designs are provided by Zierke at al. [21]. The rotor is driven
by a downstream turbine, the inflow to which is controlled by a set of variable pitch support vanes upstream of this turbine stage. Chord Reynolds numbers of the order of $3.0 \times 10^6$ were used in the experiment. A two-dimensional view of the test facility is shown in Figure 5-2, illustrating the upstream contraction and position of the stator and rotor.

![Figure 5-1: Hirep Rotor and Stator Geometry](image)

### 5.3 Rotor Thrust and Torque

Rotor shaft thrust and torque measurements were obtained using strain gauges mounted on the rotor shaft and on the rotating hub flange. Zierke et al. [21], reported that while the rotor torque measurements were accurate, the corresponding thrust measurements were subject to some experimental error due in part to limitations of the data collection apparatus and the effects of the pressure acting over the hub. Thrust and torque coefficients are reported at three advance coefficients. The resulting mean values are taken from the data-bank compiled by Zierke et al. [11].
5.4 Velocity Measurements

5.4.1 Reference Velocity

An experimental reference velocity was determined using two pressure measurements. By placing a tunnel liner pressure tap at a location 58.6% chord\(^1\) upstream of the inlet guide vanes, and a Kiel probe far upstream, the reference velocity was calculated as follows:

\[
V_{ref} = \sqrt{\frac{2}{\rho} (pT_{(ref)} - p_{ref})}
\]  

(5.1)

where \(pT_{(ref)}\) denotes the upstream total reference pressure measured using the Kiel probe and \(p_{ref}\) denotes the tunnel liner pressure tap measurement.

The reference velocity was calculated to be 35 \text{ ft/s}. This value was then used throughout the experiment, for example, in the calculation of the rotor advance coefficient:

\[
J_{hirep} = \frac{V_{ref}}{nD_{tip}}
\]

(5.2)

\(^1\)Note that the chord referred to here is the chord length of the Inlet Guide Vanes
5.4.2 Velocity Profiles

An important component of the \textit{HIREP} experimental data is the measured velocity profiles. The investigators extracted velocity profiles at three axial locations, illustrated graphically in Figure 5-4 and detailed as follows:

\textbf{Inlet Guide Vanes (IGV) Inflow} A calibrated five-hole pressure probe was used to measure both static and total pressure at a plane 37\% chord upstream of the IGV leading edge. This instrument was also used to measure axial, radial and tangential velocities at a range of points from the hub to the tunnel liner. The axial, radial and tangential velocity profiles are shown in Figure 5-3.

![Figure 5-3: Axial, radial and Tangential velocity measured at a plane 37\% upstream of the Inlet Guide Vanes](image)

\textbf{Rotor Inflow} Velocity profiles were constructed between the stator and rotor stages of the pump unit using both five-hole radial probes and Laser Doppler Velocimetry (LDV). The radial probe survey was performed at a location 49.7\% chord downstream of the IGV trailing edge while a two-component LDV survey was carried out closer to the leading edge of the rotor, at an axial location 88.5\% chord downstream of the IGV trailing edge. This two-component survey measured axial and tangential velocities.
Downstream Velocity profiles  LDV was again used downstream of the rotor at three axial locations, although a complete spanwise survey was only performed at one of these locations, corresponding to a position 32.2% chord downstream of the rotor-tip trailing edge.

![Diagram of HIREP facility and velocity profile locations from experiment]

Figure 5-4: Locations of experimental velocity profiles

5.5 Tip-gap Flow

The HIREP case provides both a quantitative and qualitative analysis of the flow in the tip-gap region. Both Cavitation and Laser Light Sheet flow visualization techniques were used by the investigators to provide a qualitative assessment of the behavior of the fluid in the tip-gap [21]. In addition, LDV measurements taken downstream of the rotor were used to evaluate the tangential velocity about the vortex axis.

Farrell and Billet [22] performed an analysis of the HIREP pump from the point of view of predicting minimum vortex pressure. Combined with the work of Zierke et al. [21], this investigation provides useful background on the nature of the tip-leakage flow for ducted propulsors / turbomachinery pumps.
5.6 Overview: Coupled Viscous / Potential Flow Analysis

A coupled analysis of the HIREP system was performed using the method described earlier in this work. With the implementation of the vortex lattice hub and duct and tip-gap model in PBD-14, this case represented the culmination of several aspects of work on the coupling process. A sectional view of the HIREP pump is shown in Figure 5.2. The high inlet-propeller plane contraction ratio (approximately 4.22 : 1) provided some difficulties from the outset.

The RANS flow solver used in conjunction with PBD-14 had previously been used almost exclusively for external flow problems. The high contraction ratio highlighted problems with the DTNS code as regards numerical compressibility and mass conservation. In an effort to circumvent such problems, three separate flow domains were investigated as the analysis progressed.

1. Equivalent Annular Pipe

2. Actual HIREP facility geometry

3. Single zone flow domain extending over region of rotor plane

The third flow domain representation mentioned above was used primarily for RANS validation and thus will not be described here.

5.6.1 Flow Domain Representation

Initially, an equivalent Annular Pipe section was used to model the HIREP facility in DTNS. This domain provided an accurate representation of the flow at the rotor plane without the added complexity associated with the full contracted geometry. This numerical representation consists of a four-zone grid, the dimensions of which are outlined in Table 5.1.

<table>
<thead>
<tr>
<th>Zone</th>
<th>Annular Pipe</th>
<th>HIREP water-tunnel</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Axial</td>
<td>Radial</td>
</tr>
<tr>
<td>Zone 1</td>
<td>81</td>
<td>73</td>
</tr>
<tr>
<td>Zone 2</td>
<td>74</td>
<td>73</td>
</tr>
<tr>
<td>Zone 3</td>
<td>74</td>
<td>73</td>
</tr>
<tr>
<td>Zone 4</td>
<td>74</td>
<td>73</td>
</tr>
</tbody>
</table>

Table 5.1: HIREP Flow Domain Discretization: Number of grid points

The annular pipe domain was used initially as the basis for the coupled analysis. Having solved the original problems regarding numerical compressibility, the actual
HIRED contracted geometry was discretized, also as a four-zone grid. The flow domain is shown in Figure 5-6.

![HIRED Annular Pipe: RANS grid](image)

Figure 5-5: HIRED equivalent annular pipe representation. As the RANS formulation is axisymmetric only the upper zones are modeled numerically - the other zones are shown to indicate the full pipe domain.

### 5.6.2 Grid Generation

In addition to the well accepted guidelines for numerical grid generation [23], both flow domain grids described above were constructed according to the following paradigms.

1. The $y^+$ parameter [13] [16], essentially a "wall" Reynolds Number based on grid cell vertical dimension, is defined as follows:

$$y^+ = u_\tau \frac{y}{\nu}$$  \hspace{1cm} (5.3)

where

$$u_\tau = \sqrt{\frac{\tau_w}{\rho U^2}}$$  \hspace{1cm} (5.4)

- $\tau_w$ is the wall shear stress
- $y$ refers to the radial dimension of the cell off the body
Based on experience with DTNS and considerations of appropriate discretization in the boundary layer, all grids generated for the current case used a $y^+$ value in the range $1 - 4$ for the first cell off the body. Subsequent cell dimensions were calculated based on a geometric progression i.e. multiplying successive cells by a common ratio to a point outside the boundary layer. This methodology ensured adequate discretization in the boundary layer, while reducing the number of overall grid points.

2. Grid spacing was constructed such that there were an adequate number of cells axially over the blade chord. Black [13] performed some convergence studies relating the number of cells axially over the blade “shadow” to rotor thrust and torque results.

3. For both the annular pipe and full domain, the zones in the region of the stator and rotor are identical in axial extent and discretization. In addition, uniform axial spacing is used for these zones (Zone 3).

4. The desired number of points in each zone was calculated such that the grids would facilitate the use of a Multigrid option in DTNS. Each zone was therefore constructed with axial and radial cell dimensions evenly divisible by eight.
5.7 Vortex Lattice Hub and Duct

The stator and rotor blades, hub and duct were represented as vortex lattice surfaces (See Figure 5-7). Since this is essentially an internal flow case, the hub and liner were modeled for half the rotor radius upstream of the rotor leading edge. The hub and duct surfaces extended approximately the same distance downstream of the trailing edge. The blended spacing routine outlined in Section 3.4 was applied to the hub and duct circumferential spacing. The optimal spacing in terms of the minimum number of circumferential elements which provided an accurate representation of the surface is tabulated in Table 5.2.

<table>
<thead>
<tr>
<th>Parameter</th>
<th># Vortex Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Duct Circumferential</td>
<td>18 (one blade interval)</td>
</tr>
<tr>
<td>Hub Circumferential</td>
<td>14 (one blade interval)</td>
</tr>
<tr>
<td>Duct Axial</td>
<td>40</td>
</tr>
<tr>
<td>Hub Axial</td>
<td>40</td>
</tr>
<tr>
<td>Blade chordwise</td>
<td>15</td>
</tr>
<tr>
<td>Blade spanwise</td>
<td>15</td>
</tr>
</tbody>
</table>

Table 5.2: HIREP Rotor Vortex Lattice Discretization

Figure 5-7: HIREP rotor vortex lattice hub and duct. Note blended circumferential spacing (portion of duct omitted for clarity)
5.8 Results and Discussion

5.8.1 Rotor Thrust and Torque

Thrust and torque coefficients for the HIREP rotor were determined at three advance coefficients following a coupled analysis using both the equivalent annular pipe representation and the actual HIREP geometry. The numerical and experimental thrust and torque coefficients are presented in Table 5.3 and Figure 5.8.

| Parameter  | Experiment | Annular Pipe | Full contracted geom.
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$K_T$</td>
<td>0.92</td>
<td>1.003</td>
<td>1.16</td>
</tr>
<tr>
<td>$10 \times K_Q$</td>
<td>0.31</td>
<td>0.3878</td>
<td>0.429</td>
</tr>
</tbody>
</table>

Table 5.3: HIREP rotor thrust and torque at $J = 2.31$

Annular Pipe Representation  The rotor thrust and torque coefficients obtained from a coupled analysis using the annular pipe flow domain over-predict the experimental values significantly. The numerical values of $K_T$ are approximately 10% higher than the experimental values at the design advance coefficient. $K_Q$ exhibits a higher discrepancy, the coupled analysis yielding a force coefficient approximately 25% over the experimental value. The numerical results were obtained using the semi-empirical tip-gap model in the coupled analysis.

HIREP Geometry  The simplified annular pipe flow domain representation was seen to over-predict the rotor thrust and torque. This was also found to be the case when the HIREP facility geometry was analyzed. However, in this case the rotor thrust and torque were even higher than the annular pipe case.

There are several factors which must be considered when comparing the numerical and experimental thrust and torque coefficients:

Analysis of Internal Flow Cases

The RANS flow solver used in the coupled analysis procedure appears ill-suited to analysis of internal flow cases. There are several issues which need to be resolved when using this flow solver for internal flow systems, namely:

- Specification of appropriate pressure boundary conditions at the far-field upstream boundary.

- Correct specification of a numerical compressibility factor to facilitate analysis of flow domains with high contraction ratios.
Figure 5-8: Coupled analysis thrust and torque coefficients for the annular pipe and full contracted geometry. Experimental results are also shown.

- Careful monitoring of pressure residuals and mass flow rates during execution to ensure convergence prior to extracting the velocity field for PBD-14.

Experimental Uncertainty

In their discussion of the experimental procedure, Zierke et al.[21] accept that the rotor thrust measurements were subject to some error, primarily due to the limitations of the experimental apparatus. In addition, a substantial error may have been introduced by failing to take account of the pressure drop across the rotor which acts across the hub face, resulting in an artificially higher measurement of rotor thrust.

A hub pressure correction factor has been calculated for the HIREP case (Appendix A). The estimated error in the reported rotor thrust is approximately 15%. When applied to the experimental data this hub pressure correction factor actually lowers the values, thus increasing the discrepancy between experimental and numerical force coefficients. However, the effects of the hub pressure, especially for a case such as HIREP which has a large hub face area, cannot be ignored.

While Zierke et al.[21] acknowledge the existence of a thrust factor acting over the hub, their results do not appear to have been adjusted to take account of this factor.
Limitations of Numerical Analysis

The existing coupled analysis process uses a vortex lattice representation of the hub and duct and includes the semi-empirical tip-gap flow model. The current analysis procedure does not, however, include the effects of blockage or thickness and viscous blade loading. These features have recently been the focus of work by Black [24] and will be incorporated into future versions of PBD.

In the case of the HIREP facility the effects of annular blockage due to blade thickness and the viscous blade loading may be significant.

5.8.2 Velocity Profiles

Velocity profiles were extracted from the converged RANS solution following the coupled analysis using both flow domain representations of the HIREP facility. The numerical velocity profiles were extracted at the same axial locations as the experimental data. These locations are illustrated in Figure 5-4.

For each of the three locations, the numerical axial and tangential velocity profiles were compared to the experimental values.

![Figure 5-9: Comparison of experimental and numerical velocity profiles 37% chord axially upstream of IGV](image)

Commencing with the inlet flow, measured at a location 37% chord upstream of the inlet guide vanes, Figure 5-9 provides a comparison between the experimental and
numerical profiles. Both the annular pipe and full domain velocity profiles show good agreement upstream of the rotor. The experimental tangential velocity profile exhibits some swirl in the inflow close to the tunnel liner. This is not evident in the numerical data as the upstream boundary condition was one of uniform axial inflow. Also, the measured axial velocity profile, non-dimensionalized using the reference velocity (Section 5.4.1) is less than unity at practically all points across the annulus. This discrepancy between the reference velocity used throughout the HIREP experiment, and the actual measured velocity profile, would suggest that the advance coefficients used in the analysis were incorrect. These advance coefficients should, in fact, be scaled in proportion to the ratio of the mean velocity from the velocity profile to the reference velocity.

![Figure 5-10: Comparison of experimental and numerical velocity profiles 49.7% chord axially downstream of IGV](image)

Figure 5-10 again provides a comparison between the experimental and numerical velocity profiles, which in this case correspond to a location 49.7% chord axially downstream of the IGV trailing edge. In effect, this profile represents the inflow to the rotor stage and as such the tangential velocity induced by the pre-swirl stator stage is clearly evident. While the experimental and numerical tangential velocity profiles agree well, the numerical axial velocity profiles exhibit a markedly different slope to the experimental profile. This may be attributed in part to the absence, at
present, of any account for the effective redistribution of the velocity profile when blockage effects are taken into consideration.

![Graph showing velocity profiles](image)

Figure 5-11: Comparison of experimental and numerical velocity profiles 32.2% chord axially downstream of rotor trailing edge

The final comparison between numerical and experimental velocity profiles is provided in Figure 5-11. These profiles represent the axial and tangential velocity at a position 32.2% chord downstream of the rotor trailing edge. The numerical velocity profiles for both the annular pipe and \textit{HIREP} geometries are almost identical. However, the presence of a finite tangential velocity in the "numerical" wake illustrates that the coupled analysis has not provided complete swirl cancellation. On the other hand, the experimental velocity profile illustrates the almost complete cancellation of the tangential velocity induced by the upstream stator. The experimental and numerical axial velocity profiles differ close to the liner. This may be due in part to the presence of a tip-vortex which may not be fully captured by the numerical analysis. In addition, the secondary flow between the blades of each stage, induced by the coupling between the tip-vortex roll-up and the passage flow, is difficult to model using the current approach.
### 5.8.3 Tip-gap Flow Modeling

With the extension of the vortex-lattice technique to model the viscous nature of the flow in the tip-gap region (Chapter 4), thrust and torque coefficients reported above included the effect of the tip-gap. The presence of the tip-gap results in a decrease in the rotor loading at the outer extremity, compared to a closed gap (See Figure 4-4). This is reflected in a reduction in rotor thrust and torque over the sealed gap case.

**Plan view of HIREP Rotor and Hub Vortex Lattice**

Figure 5-12: HIREP Rotor Tip-gap Velocity. Note the velocity vectors indicate the flow from pressure to suction side, which results in "roll-up" of the tip-vortex (Duct omitted for clarity)

The orifice flow coefficient ($C_Q$) used for the tip-gap model in the HIREP case was 0.84 Van Houten [8] has shown the tip-leakage flow to be largely unaffected by a variation of this parameter in the range $0.76 \rightarrow 0.92$. Figure 5-12 illustrates the "leakage" of fluid from the blade pressure side to the suction side. The velocity vectors shown in this figure correspond to the velocity at the gap control points.

### 5.9 Summary

Coupled analysis of the HIREP case using two separate flow domain representations results in rotor thrust and torque coefficients which are significantly higher than experimentally reported values. A number of factors, both experimental and numerical, have been identified which may help explain the discrepancy between the results.
Chapter 6

Case II : NSMB Ka4-55 Ducted Propulsor

6.1 Overview

In addition to the HIREP case outlined in the preceding chapter, a Netherlands Ship Model Basin (NSMB), Kaplan series ducted propulsor (designated Ka4-55) was used as a validation case for the vortex lattice representation of the hub and duct in PBD-14. In addition, this test case provided a means of validating the semi-empirical tip-gap model.

Van Manen [25] provides a comprehensive overview of the effect of radial load distribution on the performance of ducted propulsors. The experimental data used in this section corresponds to one element of a matrix of propeller-nozzle combinations tested by Van Manen. The Ka4-55 propeller has minimal rake and skew, a pitch-to-diameter ratio, $P/D$, of unity, a parabolic mean line and NACA 4-digit thickness distribution. The nozzle used in conjunction with this propeller was Nozzle 19.

Nozzle 19 [26][6] was constructed using a NACA 250 mean line and a NACA 0015 thickness distribution [27]. The duct angle of attack is 10.2° and the design advance coefficient for this propulsor is 0.36. The tip-gap, expressed as a percentage of propeller diameter ($D$) is 0.42%.

The available experimental data is in the form of separate rotor and nozzle thrust and torque measurements over a range of advance coefficients. This data was also used by Van Houten [8] to validate a similar method of representing the duct in potential flow using a vortex lattice.

The primary objective of this section is to provide a means of validating the vortex lattice lifting surface enhancements described earlier in this work. In addition, this validation case is presented as a step-by-step guide to the process of building a propeller analysis model from scratch. The description will therefore follow the flowchart
illustrated in Figure 2-2.

6.2 Hull Problem: Setup

A converged propeller inflow was obtained using the RANS flow solver. This finite difference numerical technique requires the discretization of the flow domain as a two-dimensional grid of points (in \( x, r \) coordinates). Since the flow is axisymmetric a full three dimensional representation of the flow domain is not required. Depending on the specific problem, it may be appropriate to model the entire underwater vehicle, as in the case of submersibles. Alternatively, it may be sufficient to represent only the stern of the vehicle in some cases [1].

6.2.1 Grid Generation

The \( Ka_4-55 \) propeller was tested in open water and therefore the flow domain was extended from the propeller shaft to a radial boundary four propeller diameters from the shaft centerline, as illustrated in Figure 6-1. The axial limits of the flow domain extend from four propeller diameters upstream of the duct leading edge to a position three diameters downstream of the duct trailing edge (See Figure 6-1). For this specific configuration the duct length is equivalent to one rotor radius.

Using an existing two-dimensional structured grid generation program, INMESH [28], the flow domain was initially subdivided into six zones or blocks, a consequence of the constraint that only one boundary condition (for example NOSLIP) can be specified along any one side of a zone. A pre-processing program, developed at the MIT Marine Hydrodynamics Lab [29], was modified by the author to facilitate rapid generation of a six-zone grid about a duct. The resulting discretization, in terms of cell density, is illustrated in Table 6.1.

<table>
<thead>
<tr>
<th>Zone</th>
<th>Axial</th>
<th>Radial</th>
</tr>
</thead>
<tbody>
<tr>
<td>Zone 1</td>
<td>81</td>
<td>113</td>
</tr>
<tr>
<td>Zone 2</td>
<td>66</td>
<td>113</td>
</tr>
<tr>
<td>Zone 3</td>
<td>74</td>
<td>113</td>
</tr>
<tr>
<td>Zone 4</td>
<td>81</td>
<td>81</td>
</tr>
<tr>
<td>Zone 5</td>
<td>66</td>
<td>81</td>
</tr>
<tr>
<td>Zone 6</td>
<td>74</td>
<td>81</td>
</tr>
</tbody>
</table>

Table 6.1: RANS grid discretization of \( Ka_4-55 \) flow domain
Figure 6-1: Computational grid for RANS modeling of flow domain for Ka-55 propeller. The main figure illustrates the decomposition of the flow field into several blocks or zones. The upper left inset figure illustrates the discretization over the axial and radial extent of the rotor (not shown). The final inset figure shows leading edge grid details.

6.2.2 RANS Converged Solution

Having provided an appropriate numerical representation of the flow domain, the RANS flow solver was then used to obtain a converged flow solution for the axisymmetric problem. At this stage of the coupling process, the flow solver does not account for the presence of the propeller.

Axial velocity contours for the converged flow field are illustrated in Figure 6-2. Note that in the absence of the propeller, and due to the relatively high duct-angle of attack (10.2°), the flow separates off the trailing quarter-chord of the upper surface of the duct.

6.3 Propeller Problem: Setup

The potential flow propeller design and analysis code requires the blade geometry to be input as a B-spline net. This data can be extracted from CAD/CAM packages or can be derived from designer data pertaining to the blade geometry.
The B-spline representation of Ka4-55 was obtained by fitting a set of Cartesian coordinates with a B-spline surface [30]. An initial propeller inflow field was extracted from the converged RANS solution. The hub and duct streamlines are used as the basis for the vortex lattice representation of the hub and duct. It is important to note that PBD models the duct inner surface and not the duct mean camber surface. The Ka4-55 blade, hub and duct geometry is illustrated in Figure 6-3.

6.4 Coupled Analysis

Having performed the initial propeller analysis in PBD-14, the propeller forces were overlaid on the computational grid and the iterative coupling procedure was initiated. A coupled analysis was performed at four Advance coefficients, $J = 0.24, 0.36, 0.544, 0.72$. The design advance coefficient for this propulsor corresponds to $J = 0.36$. At each data point the coupled analysis was re-iterated until the thrust and torque coefficients had converged. A sample convergence profile, illustrating the variation of $K_T$ and $K_Q$ with increasing coupling cycles, is shown in Figure 6-4.
Figure 6-3: Ka4-55 Blade, hub and duct vortex lattice. The blade is input to PBD as a B-spline control polygon. The hub and duct vortex-lattice surfaces are discretized using the hub and duct streamlines from the RANS flow field.

In the example shown in Figure 6-4, the coupling process cycled fifteen times between the RANS flow solver and the vortex lattice propeller design/analysis code. The ultimate converged flow field is shown in Figure 6-5. Comparing this figure with the converged flow field in the absence of the propeller (Figure 6-2) the effect of the propeller induction is apparent: the flow now remains largely attached to the suction side of the duct and the presence of the propeller results in the contraction of the streamlines entering the propeller plane.

6.5 Discussion of Results

6.5.1 Rotor Thrust and Torque

Rotor thrust and torque coefficients obtained following a coupled analysis of the Ka4-55 propulsor are presented in Figure 6-6. This figure also illustrates the experimentally measured thrust and torque at the same advance coefficients, taken from Van Manen [25]. The numerical result compares well with the experimental data at the design advance coefficient. Note that for the results presented in Table 6.2, the effects of the tip-gap have not been considered and hence the actual thrust and torque coefficients will be slightly lower than those presented therein.

The larger discrepancies at off-design operating conditions may be due in part to the absence of viscous coupling effects [24] in the current version of PBD-14.
Figure 6-4: Variation of $K_T$ and $K_Q$ with coupling cycles. Each symbol represents an analysis data point using $PBD-14$. Advance coefficient is 0.36.

Also, the blade transition wake is currently aligned with the circumferential mean velocity. As yet, iterative blade wake alignment has not been implemented and this will certainly play an important role at off-design operating points.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.24</td>
<td>0.252</td>
<td>0.410</td>
<td>0.2484</td>
<td>0.4223</td>
</tr>
<tr>
<td>0.36</td>
<td>0.245</td>
<td>0.400</td>
<td>0.2380</td>
<td>0.4083</td>
</tr>
<tr>
<td>0.544</td>
<td>0.206</td>
<td>0.344</td>
<td>0.2084</td>
<td>0.3680</td>
</tr>
<tr>
<td>0.72</td>
<td>0.143</td>
<td>0.250</td>
<td>0.1638</td>
<td>0.3046</td>
</tr>
</tbody>
</table>

Table 6.2: Comparison of experimental and numerical thrust and torque coefficients for the $Ka4-55$ case

6.5.2 Tip-gap Flow Modeling

The tip-gap to rotor diameter ratio for the $Ka4-55$ propulsor was 0.42%. Even this small tip-gap dimension results in a reduction in load at the blade tip compared to the sealed gap. A comparison of rotor spanwise circulation distributions for finite and zero gap dimensions is presented in Figure 6-7. This particular circulation distribution was extracted from a coupled analysis at $J = 0.72$. When applied to the thrust and torque coefficients presented in Figure 6-6, this change in force coefficient will bring
Figure 6-5: Converged RANS flow field following iteration of the coupling process. The presence of the rotor is modeled by body forces in the RANS domain. The presence of the propeller has altered the flow over the duct such that flow now remains attached (Compare with Figure 6-2)

the torque curve closer to experimental values, however, the effect on $K_T$ will vary depending on advance coefficient.

6.6 Summary

The 114-55 propulsor provided a useful test case for the vortex-lattice lifting-surface enhancements. The coupled analysis thrust and torque numerical results show good agreement with experimental data, except at advance coefficients far from the design operating point. However, this validation case warrants further investigation, possibly at a point when blade wake alignment and viscous load coupling have been integrated fully into PBD-14.
Figure 6-6: Comparison of numerical and experimental thrust and torque coefficients for the K4-55 propeller in Nozzle 19 over a range of advance coefficients.

Figure 6-7: Comparison of radial distribution of circulation for zero and finite tip-gaps.
Chapter 7

Conclusion

7.1 Vortex Lattice Lifting Surface Code Enhancements

Coupled analysis of complex single- or multi-stage marine propulsors combines the elegance and computational efficiency of the vortex lattice method, with aspects of the flow behavior with can only be captured using viscous schemes. This coupling procedure has previously been shown to be both robust and computationally efficient.

The current work, therefore, has attempted to enhance the existing vortex-lattice technique such that the coupled analysis will provide an even more accurate model of the propulsor. These lifting surface enhancements have focused on two areas: (1) The implementation of a vortex lattice hub and duct and (2) implementation of a semi-empirical tip-gap flow model.

7.1.1 Vortex Lattice Hub and Duct

By imposing the Kinematic Boundary Condition directly on the hub and duct, the vortex lattice representation of these surfaces supersedes the method of images as the preferred route to hub, duct modeling. Currently, the use of a vortex lattice hub and duct is restricted to propeller analysis, however, the ultimate objective is to also replace the method of images in design mode.

The vortex lattice hub and duct added to the complexity of the overall PBD-14 formulation, however, by utilizing accelerated matrix solution algorithms, even relatively large systems (26 x 26 elements on the blade with comparable hub / duct discretization) did not prove computationally prohibitive. The increased memory allocation required with these extended systems, however, may in the short term prove to be a limiting factor in the size of system that can be analyzed. For this reason, it has been difficult to perform adequate convergence studies using the enhanced representation.
7.1.2 Numerical Tip-gap Flow Modeling

The implementation of the tip-gap flow model has provided a means of capturing the viscous nature of the flow in the tip clearance region within a potential flow code, albeit using an empirically derived co-efficient. The reduction in blade loading when the gap is considered is substantial, even in the case of very small tip gap dimensions (of the order of 0.2% of propeller diameter).

7.2 Concurrent Work on PBD-14

As the title reads, this thesis has focused on advances in ducted propulsor analysis using vortex lattice techniques. Significant advances have been made in other areas of propeller vortex lattice methods at the MIT Marine Hydrodynamics Facility.

The following features have recently been implemented in PBD-14 by Black [24]:

1. Thickness Load Coupling
2. Viscous Load Coupling
3. Blockage effects

In future versions of PBD these enhancements will be integrated with the work described in this thesis. The inclusion of the effects of blockage / thickness load coupling / viscous load coupling / tip-gap and vortex lattice hub and duct in a single propulsor design and analysis code will ensure a more comprehensive technique.

7.3 Future Vortex-Lattice Enhancements

7.3.1 Vortex Lattice Hub / Duct in Design Mode

Currently, the use of the vortex lattice hub and duct is restricted to propulsor analysis. Ultimately, the vortex lattice hub and duct will be used for both the design and analysis processes. Existing propeller design uses the method of images to model the hub and duct.

Designing a ducted propulsor using vortex lattice representation of the hub and duct will require one of two approaches:

Specified duct circulation The current design procedure in PBD-14 requires the user to input a radial distribution of circulation on the blade. By requiring a similar distribution on the duct (the effect of the hub is small in comparison) the system could be solved in the usual manner. This initial duct circulation could be provided by an external program, such as DPLL [14].
Iterative solution procedure By assuming an initial circulation distribution on the duct the problem could be approached from an iterative standpoint. This formulation would in effect be a pseudo-design method whereby elements of the analysis procedure could be used to determine the appropriate duct circulation.

7.3.2 Blade Wake Alignment
Currently, the trailing vortices which constitute the blade transition wake, are convected downstream with the circumferential mean induced velocity. Therefore the wake is not currently aligned with the local velocity in the wake. This may provide an explanation for the differences between experimental and numerical results for the Ka4-55 case at off-design operating conditions.
Wake-alignment is proposed for future versions of PBD-14.

7.3.3 Tip-fin Modeling
The current implementation of the vortex lattice hub and duct facilitates symmetric tip-fin geometries. Essentially, the geometry for such cases is generated by specifying an angular increment over which the blade tip fin will extend. Current restrictions prohibit the analysis of asymmetric geometries and tip-fins which are not aligned with the blade tip-chord pitch angle.

It is envisaged that future versions of PBD will facilitate analysis of arbitrary symmetric or asymmetric tip-fin geometries. A component of this implementation will include the inclusion of a band (or fin) force calculation routine. At present, only the blade forces are calculated in PBD-14, the duct forces can be extracted from a pressure integration of the duct boundary points in the RANS solution, if desired.

In addition to the above, current interest in the analysis of complex tip-geometry rotors, within a duct (as illustrated in Figure 7-1), may warrant the inclusion of such analysis capabilities within the coupled analysis. Such a formulation is currently being investigated.

7.4 Internal Flows
The global increase in demand for high-speed surface craft for both commercial and military applications has necessitated the application of propeller design and analysis tools to water-jet propulsors. Essentially, these propulsors are similar in many ways to the HIREP case examined in this work - multi-stage; internal flow systems. The
Figure 7-1: Modified MIT Sirenian rotor design incorporating complex tip fin geometry inside a duct!

HIREP coupled analysis has provided the foundation for future work in the design and analysis of water-jet propulsion systems. A critical aspect of correctly modeling internal flows concerns the "reliability" of the RANS flow solver in terms of robustness and accuracy.

7.5 Coupled Analysis: Closing Remarks

The coupled analysis procedure outlined in this work remains the technique of choice for the design and analysis of single- or multi-stage marine propulsors, in terms of computational efficiency, robustness and relative simplicity of the formulation. Future enhancements will further reinforce the technique.

The HIREP case highlighted some of the problems associated with modeling an internal flow system. Further validation of this case is planned as a means of gaining more experience for future pump-jet application studies.
Appendix A

Hub Pressure Correction

This note provides an analysis of the hub correction factor applied to the HIREP geometry. The correction factor derivation follows closely that derived by Kerwin [31] except for some non-dimensionalization issues.

![Diagram of Hub Pressure Probe Measurement Locations]

Figure A-1: HIREP radial pressure probe measurement locations

A.1 Review of Derivation

The thrust correction factor due to the pressure on the hub face can be non-dimensionalized as follows:

\[ C_F = \frac{F}{\frac{1}{2} \rho V_s^2 A^*} \]  \hspace{1cm} (A.1)
where \( A^* \) denotes some arbitrary area.

The Pressure Coefficient is defined as follows:

\[
C_p = \frac{p - p_\infty}{\frac{1}{2} \rho V_s^2}
\]  

(A.2)

Define the pressure coefficient on body at hub gap as \( C_{p0} \). The force on the hub face is given as follows:

\[
F_0 = \int \int_S p \cdot \hat{n} \, ds
\]

(A.3)

In the case of the hub pressure this integral is formulated as follows when the hub is stationary:

\[
F_0 = \int_{r_i}^{r_o} p_\infty 2\pi r \, dr
\]

(A.4)

Rotating Hub:

\[
\delta F = F - F_0 = \int_{r_i}^{r_o} (p - p_\infty) 2\pi r \, dr
\]

(A.5)

By formulating the equations of motion of the system:

\[
\frac{1}{\rho} \frac{\partial p}{\partial r} = \frac{V_i^2}{r} = \frac{V_{10}^2 r}{r_o^2}
\]

(A.6)

After some algebra:

\[
p - p_\infty = \frac{1}{2} \rho V_{10}^2 [\left(\frac{r}{r_o}\right)^2 - 1]
\]

(A.7)

On the body:

\[
C_{p0} = \frac{p - p_\infty}{\frac{1}{2} \rho V_s^2}
\]

(A.8)

and

\[
p - p_\infty = (p_o - p_\infty) + (p - p_\infty)
\]

(A.9)

\[
p - p_\infty = \frac{1}{2} \rho V_s^2 C_{p0} + \frac{1}{2} \rho \left(\frac{\omega r_o}{2}\right)^2 [\left(\frac{r}{r_o}\right)^2 - 1]
\]

(A.10)

\[
\delta F = \int_{r_i}^{r_o} \frac{1}{2} \rho V_s^2 C_{p0} 2\pi r \, dr + \int_{r_i}^{r_o} \frac{1}{2} \rho \left(\frac{\omega r_o}{2}\right)^2 [\left(\frac{r}{r_o}\right)^2 - 1] 2\pi r \, dr
\]

(A.11)

The resulting change in Thrust Coefficient is determined as follows:

\[
\delta C_T = \frac{\delta F}{\frac{1}{2} \rho V_s^2 A^*}
\]

(A.12)

\[
\delta C_T = \frac{C_{p0} \pi}{A^*} [r_o^2 - r_i^2] - \frac{1}{8} \pi \frac{C_{p0}}{r_o^2 A^*} \left(\frac{\omega V_s}{r_o^2}\right)^2 [r_o^2 - r_i^2]^2
\]

(A.13)
A.2  **HIREP** Correction factor

With reference to Figure 1 above the added force will be the difference between the contribution at A and B. In the case of a non-tapered hub ( \( r_o, r_i \) constant at A and B ) the second term in Equation A.13 above is constant. These components at A and B cancel out!

Represent the change in pressure coefficient across the rotor as \( \Delta C_{po} \).

\[
\Delta C_{po} = C_p|_B - C_p|_A \tag{A.14}
\]

Therefore equation 13 above reduces to:

\[
\delta C_T = \frac{\Delta C_{po} \pi}{A^*}[r_o^2 - r_i^2] \tag{A.15}
\]

For **HIREP** the non-dimensional area is the Annular area:

\[
A^* = \pi[r_r^2 - r_o^2] \tag{A.16}
\]

\[
\delta C_T = \frac{\Delta C_{po}}{[r_r^2 - r_o^2]}[r_o^2 - r_i^2] \tag{A.17}
\]

\[
\delta C_T = \frac{\Delta C_{po}}{[1 - \frac{r_i^2}{r_o^2}]\left[(\frac{r_o}{r_r})^2 - (\frac{r_i}{r_r})^2\right]} \tag{A.18}
\]

\[
\frac{r_o}{r_r}\big|_{hirep} = 0.5 \tag{A.19}
\]

\[
\frac{r_i}{r_r}\big|_{hirep} \simeq 0.1 \tag{A.20}
\]

\[
\delta C_T = 0.32\Delta C_{po} \tag{A.21}
\]

From **HIREP** report:

\[
K_T = \frac{F_T}{\rho m^2 D_{tip}^4} \tag{A.22}
\]

\[
C_T = \frac{F_T}{\frac{1}{2}\rho V_{ref}^2 A} \tag{A.23}
\]

Therefore:

\[
C_T = \frac{K_T \rho m^2 D_{tip}^4}{\frac{1}{2}\rho V_{ref}^2 A} \tag{A.24}
\]

- \( V_{ref} \), the Reference Axial Velocity at a point 58.6 percent chord upstream of the Inlet Guide Vanes ( IGV ).
- \( A \) is the annular area through the contraction = 7.22 feet
- \( D_{tip} \) = Rotor Tip Diameter = 3.5 feet
\[ C_T = \frac{2K_T n^2 D_{tip}^4}{V_{ref}^2 A} \]  
\[ J = \frac{V_{ref}}{n D_{tip}} \]  
(A.25)  
(A.26)

Therefore:
\[ C_T = \frac{2K_T D_{tip}^2}{J^2 A} \]  
\[ C_T = \frac{8K_T}{J^2 \pi [1 - (\frac{D_{huk}}{D_{tip}})^2]} \]  
(A.27)  
(A.28)

But \( \frac{D_{huk}}{D_{tip}} |_{HIREP} = 0.5 \).
\[ C_T = \frac{3.3953K_T}{J^2} \]  
(A.29)

At the design point: \( J = 2.31 \)
\[ C_T |_{J=2.31} = 0.63629K_T \]  
(A.30)

Values of \( C_p \) given in \textit{HIREP} report:
\[ \Delta C_p |_{BA} \simeq 0.27 \]  
(A.31)
\[ \delta C_T = (0.32)\Delta C_p \]  
(A.32)
\[ \delta C_T |_{J=2.31} = 0.0864 \]  
(A.33)
\[ \delta K_T |_{J=2.31} = 0.1358 \]  
(A.34)

**A.3 HIREP Experimental Values of \( K_t, K_q \)**

The following values are quoted in the \textit{HIREP} report [21]:
\[ K_T^{expt} |_{J=2.31} = 0.92 \]  
(A.35)
\[ K_T^{design} |_{J=2.31} = 0.77 \]  
(A.36)

Thrust co-efficient values determined during the experiments are approximately 16\% higher than the design values at the same Advance Coefficient. Applying the thrust correction factor to the Design value of \( K_t \):
\[ K_T^{+hubpress} |_{J=2.31} = 0.91 \]  
(A.37)
Appendix B

Additional Validation Cases

B.1 Overview

This section presents some additional cases used to validate the coupled RANS-potential flow propeller design and analysis method. In each case a brief overview of the propulsor characteristics is provided, as is a summary of the features of the propulsor which best illustrate the improvements made to the propeller design and analysis code. The additional validation cases are as follows:

1. Propeller 4119
2. Propeller 4497 / MIT-PRESWIRL
3. NSMB R4-55
### B.2 Propeller 4119

Propeller 4119 is a three-bladed open propeller designed by Denny [32] for operation in uniform inflow. More comprehensive propeller characteristics are presented in the tables below. The Radial distribution of Circulation and Force Coefficients for this propeller were measured experimentally by Jessup [33].

Diameter, \( D = 1.00 \text{ ft.} \ (0.305 \text{ m}) \\
Rotation: Right Hand \\
Number of Blades: 3 \\
Hub-Diameter Ratio: 0.20 \\
Skew \( \theta_s \), Rake: None \\
Design Advance Coefficient, \( J \): 0.833 \\
Section Thickness Form: NACA66 (DTRC Modified) \\
Section Meanline, NACA, \( a=0.8 \) \\
Design Thrust Coefficient, \( K_T \): 0.150

<table>
<thead>
<tr>
<th>( r/R )</th>
<th>( C/D )</th>
<th>( P/D )</th>
<th>( \theta_s )</th>
<th>( t_T/D )</th>
<th>( t_M/C )</th>
<th>( f_M/C )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2</td>
<td>0.3200</td>
<td>1.105</td>
<td>0</td>
<td>0</td>
<td>0.2055</td>
<td>0.01429</td>
</tr>
<tr>
<td>0.3</td>
<td>0.3625</td>
<td>1.102</td>
<td>0</td>
<td>0</td>
<td>0.1553</td>
<td>0.02318</td>
</tr>
<tr>
<td>0.4</td>
<td>0.4048</td>
<td>1.098</td>
<td>0</td>
<td>0</td>
<td>0.1180</td>
<td>0.02303</td>
</tr>
<tr>
<td>0.5</td>
<td>0.4392</td>
<td>1.093</td>
<td>0</td>
<td>0</td>
<td>0.09016</td>
<td>0.02182</td>
</tr>
<tr>
<td>0.6</td>
<td>0.4610</td>
<td>1.088</td>
<td>0</td>
<td>0</td>
<td>0.06960</td>
<td>0.02072</td>
</tr>
<tr>
<td>0.7</td>
<td>0.4622</td>
<td>1.084</td>
<td>0</td>
<td>0</td>
<td>0.05418</td>
<td>0.02003</td>
</tr>
<tr>
<td>0.8</td>
<td>0.4347</td>
<td>1.081</td>
<td>0</td>
<td>0</td>
<td>0.04206</td>
<td>0.01967</td>
</tr>
<tr>
<td>0.9</td>
<td>0.3613</td>
<td>1.079</td>
<td>0</td>
<td>0</td>
<td>0.03321</td>
<td>0.01817</td>
</tr>
<tr>
<td>0.95</td>
<td>0.2775</td>
<td>1.077</td>
<td>0</td>
<td>0</td>
<td>0.03228</td>
<td>0.01631</td>
</tr>
<tr>
<td>1.0</td>
<td>0.0900</td>
<td>1.075</td>
<td>0</td>
<td>0</td>
<td>0.03160</td>
<td>0.01175</td>
</tr>
</tbody>
</table>

Table B.1: Geometry of DTRC Propeller 4119.

### B.2.1 Features

The DTMB 4119 case was analyzed using both a vortex-lattice hub and the method of images to represent the hub. At the design advance coefficient ( \( J = 0.833 \) ), the solved spanwise distribution of circulation on the blade was extracted for each case (Figure B-2). The thrust and torque coefficients were also calculated.

For this specific case, the circulation distributions from both hub models vary only slightly over a portion of the blade near the hub (Figure B-2). This results in a difference of approximately 1\% in the thrust coefficients between the two different hub models. Similarly, the torque coefficient exhibits a decrease of approximately 0.5\% when the hub image method is used. The results are shown in Table B.2.

While calculated force coefficients for propeller 4119 exhibit only minor variation when the more accurate vortex-lattice hub model is used, more significant discrep-
Figure B-1: DTRC Propeller 4119

Ancies are anticipated when analyzing thicker blade sections and more highly loaded propellers.

<table>
<thead>
<tr>
<th></th>
<th>$K_T$</th>
<th>$10 \times K_Q$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Image Hub</td>
<td>0.1524</td>
<td>0.2794</td>
</tr>
<tr>
<td>Vortex Lattice Hub</td>
<td>0.1540</td>
<td>0.2811</td>
</tr>
<tr>
<td>Experiment</td>
<td>0.146</td>
<td>0.280</td>
</tr>
</tbody>
</table>

Table B.2: Comparison of DTRC 4119 thrust and torque coefficients at design advance coefficient: $J = 0.833$
Figure B-2: Propeller 4119: Comparison of spanwise solved circulation distribution using both methods of hub representation at $J = 0.833$
B.3 Propeller 4497 / MIT-PRESWIRL

Forces on a stator and rotor combination were obtained experimentally in the MIT water tunnel to validation the coupled analysis method, again using the two hub representation techniques. The stator was designed by Bowling [34] to provide swirled inflow to David Taylor Model Basin model Propeller 4497 ( Figure B-3 ). The stator geometry and rotor geometry are described by Bowling [34].

![Figure B-3: DTRC Propeller 4497](image)

B.3.1 Features

The coupled analysis was performed using a computational grid which represented a cylindrical water-tunnel with cross sectional area equivalent to the square cross-section MIT facility, where the propulsor was tested. The three zone grid is illustrated in Figure B-4.

For the purposes of validating the vortex lattice hub technique, the coupled analysis considered only the rotor. This was useful as it simplified the analysis while still permitting comparison with thrust and torque coefficients for the rotor only reported by Bowling [34]. These force coefficients were compared for the case of both the hub
image method and the vortex lattice hub. In addition, each analysis was performed with and without blade thickness considered. Numerical and experimental results are presented in Figure B-5. A more detailed analysis of the stator-rotor combination is provided by Kerwin et al.[5].
Figure B-5: Convergence of thrust and torque coefficients for both hub models: hub images and vortex lattice hub; with and without blade thickness
B.4 Propeller R4-55

Van Houten [8] used a banded propeller, NSMBR4-55 (Figure B-6), as a basis for the validation of the vortex lattice Lifting Surface Code, DPSF. This is a four-bladed banded propeller with zero rake and skew. Van Houten also provides some of the initial experimental data on this propeller reported by Van Gunstern (in [8]). The propeller and band geometry is tabulated in Tables B.3 and B.4.

<table>
<thead>
<tr>
<th>r/R</th>
<th>C/D</th>
<th>P/D</th>
<th>θₐ</th>
<th>tₑ/D</th>
<th>tₑ/D</th>
<th>tₑ/D</th>
<th>fₑ/C</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.182</td>
<td>0.179</td>
<td>1.423</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0.039</td>
</tr>
<tr>
<td>0.3</td>
<td>0.208</td>
<td>1.402</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0.031</td>
</tr>
<tr>
<td>0.4</td>
<td>0.232</td>
<td>1.389</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0.0215</td>
</tr>
<tr>
<td>0.5</td>
<td>0.254</td>
<td>1.380</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0.016</td>
</tr>
<tr>
<td>0.6</td>
<td>0.273</td>
<td>1.379</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0.015</td>
</tr>
<tr>
<td>0.7</td>
<td>0.288</td>
<td>1.386</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0.015</td>
</tr>
<tr>
<td>0.8</td>
<td>0.299</td>
<td>1.408</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0.015</td>
</tr>
<tr>
<td>0.9</td>
<td>0.306</td>
<td>1.446</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0.015</td>
</tr>
<tr>
<td>0.95</td>
<td>0.308</td>
<td>1.472</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0.015</td>
</tr>
<tr>
<td>1.0</td>
<td>0.309</td>
<td>1.502</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0.015</td>
</tr>
</tbody>
</table>

Table B.3: Geometry of NSMB Propeller R4-55.

<table>
<thead>
<tr>
<th>Type</th>
<th>Thickness</th>
<th>C/D</th>
<th>fₑ/C</th>
<th>tₑ/D</th>
<th>α</th>
</tr>
</thead>
<tbody>
<tr>
<td>NACA 259</td>
<td>0.024</td>
<td>0.1474</td>
<td>0.0853</td>
<td>0.0354</td>
<td>13.17⁰</td>
</tr>
</tbody>
</table>

Table B.4: R4-55 Band Geometry

B.4.1 Features

The NSMBR4-55 banded propeller was used in the initial stages of vortex lattice hub and duct development in an attempt to compare the solved circulation distribution obtained using this duct representation with experimental or other numerical data. As such, this analysis was performed using a uniform inflow field i.e. a coupled analysis of this propeller was not performed.

Comparing the solved circulation obtained with PBD-14 and Van Houten’s program, DPSF, the two circulation distributions agree well. Note that Van Houten’s investigation did not include a hub model, hence the drop in load at the inner radii (See Figure B-7). Thrust and torque coefficients could not be compared as PBD-14 does not, as yet, include a force calculation routine for the thrust and torque on the band. Such a scheme will be implemented in the near future.
Figure B-6: NSMB Propeller R4-55

Figure B-7: Comparison of circulation distributions for NSMB R4-55 propeller in uniform inflow; PBD-14 versus results by Van Houten. The PBD circulation distribution using the method of images is also shown.
Bibliography


"Vortex-Lattice" and "Lifting-Surface" are two different words in degree bk (rehyphenated).

NAME VARIATES: ☐

IMPRINT (COPYRIGHT)

COLLATION: 892

ADD. DEGREE:   DEPT.:   

SUPERVISORS:   

NOTES:   

cat'r:  date:  

DEPT  O.E.  page:  537

YEAR  1997  DEGREE  Nav.E

NAME  MCHUGH, Gerard Paul