Preliminary Design and Analysis of Propulsors for Axisymmetric Underwater Vehicles

by

Todd E. Taylor

A.B., Biochemistry, Bowdoin College (1990)
M.S., Ocean Engineering,
Massachusetts Institute of Technology (1994)

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

Doctor of Philosophy in Hydrodynamics

at the

MASSACHUSETTS INSTITUTE OF TECHNOLOGY

September 1996

© Massachusetts Institute of Technology 1996. All rights reserved.

Author .......................................................... Department of Ocean Engineering

September 4, 1996

Certified by ..................................................

Justin E. Kerwin
Professor of Naval Architecture
Thesis Supervisor

Accepted by ..................................................

A. Douglas Carmichael
Chairman, Departmental Committee on Graduate Students

MASSACHUSETTS INSTITUTE
OF TECHNOLOGY

FEB 10 1997

LIBRARIES
Preliminary Design and Analysis of Propulsors for Axisymmetric Underwater Vehicles

by

Todd E. Taylor

Submitted to the Department of Ocean Engineering on September 4, 1996, in partial fulfillment of the requirements for the degree of Doctor of Philosophy in Hydrodynamics

Abstract

Hydrodynamic design of marine propellers is accomplished in three steps. The first step, preliminary design, is traditionally the calculation of optimum radial force distributions. The second step is the determination of a blade surface which produces the optimum force distribution. The third step is an analysis of the designed blade surface. A technique called PBD-14/Design already exists for the second step of the design process via a vortex-lattice lifting-surface design technique. PBD-14/Design is capable of modeling contracted streamtube geometries. The purpose of the current work was to improve the design process for ducted propulsors on highly tapered afterbodies by creating a new preliminary design technique and a new analysis technique.

In the present work, a lifting-line model was developed for the preliminary design of single or multiple blade-row ducted propulsors on axisymmetric underwater vehicles with highly-tapered afterbodies. This model, called DPLL, was developed to improve preliminary design information needed by design techniques like PBD-14/Design. Current preliminary design methods either do not treat contracted streamtube geometries or do not completely model the effect of the duct.

The ideal circulation distribution on multiple blade-row ducted propulsors is generally known. DPLL allows a user specification of the form of the induced tangential velocity on each blade-row of the propulsor. In this way, the designer can work to identify a distribution of induced tangential velocity that is acceptable for later steps in the design process while also matching a desired circulation distribution.

Rather than optimize individual pieces of the propulsor, DPLL models all of the components of a highly tapered propulsor, including the afterbody surface. Due to the close proximity of the duct, blade-rows, and afterbody in this type of propulsor, it is important for the designer to have a tool that models the interaction of these components. DPLL performs a duct mean-camber design to allow the designer to investigate the effect of variations in duct loading and shape. The effect of variations in vehicle afterbody taper can also be examined.

Also in the present work, a vortex-lattice lifting-surface analysis partner, PBD-14/Analysis, was created for the PBD-14/Design program. This design and analysis pair allows for the blade design and analysis of single and multiple blade-row open and ducted marine propulsors, including a treatment of contracted streamtube geometries. Until the development of PBD-14/Analysis, blade geometries designed in PBD-14/Design were typically analyzed with separate analysis programs which treat contracted streamtube geometries only in a rudimentary fashion. Also, it was necessary to convert the output files from PBD-14/Design into the very different input file format of the separate analysis codes.

The new combined design-analysis format of PBD-14 allows for quick analysis of blade designs from PBD-14/Design. Like PBD-14/Design, the PBD-14/Analysis was developed to function with an input effective inflow to the propulsor or to perform the analysis coupled with an axisymmetric
viscous flow solver.

The new preliminary design tool \textit{DPLL}, the pre-existing \textit{PBD-14/Design}, and the new \textit{PBD-14/Analysis} technique are all capable of properly treating propulsors with arbitrary streamtube geometry. When used together, these techniques provide the major steps to a design process for multiple blade-row ducted propulsors tailored to operate on axisymmetric underwater vehicles with high afterbody taper.

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

My sincere appreciation goes to Professor Justin Kerwin for his good-humored and endless guidance throughout my graduate work.

I am grateful for the assistance provided by my thesis committee members: Professor A.D. Carmichael, Professor Mark Drela, Dr. David Keenan, and Professor Justin Kerwin.

Thanks to Scott Black and Gerard McHugh for many helpful discussions and their time spent performing RANS computations.

Thanks to Dianne Egnor for her help with making some of the figures look right and figuring out LATEX problems.

I was grateful to have Beth Lurie to work alongside me and with me during graduate school.

Finally, thanks to my family and Ricky for always encouraging me to work hard but have fun.

A portion of my graduate education was supported under a National Defense Science and Engineering Graduate Fellowship (NDSEG). This thesis was supported in part by the Office of Naval Research and the David Taylor Model Basin under Grant N00014-95-1-0389 and under a Department of Defense Augmentation Award for Science and Engineering Research Training (AASERT) through ONR Grant N00014-95-1-1240.
Contents

1 Introduction ............................................. 20

2 A Lifting-Line Theory for Ducted Propulsors on Highly Tapered
  Afterbodies ............................................. 26
  2.1 Background ............................................. 26
     2.1.1 A need for generalized streamtube geometries ....... 27
     2.1.2 Improved propeller-induced velocity distributions ... 27
  2.2 Present technique overview ................................ 28
  2.3 Duct representation ....................................... 30
  2.4 Vehicle body representation ................................ 32
  2.5 Lifting line blade-rows ..................................... 34
     2.5.1 Geometry ........................................... 34
     2.5.2 Advance coefficient ................................... 35
  2.6 Radial distribution of circulation and
     induced velocities ........................................... 37
     2.6.1 Representation of circulation on the lifting line ....... 37
     2.6.2 Design circulation distributions ....................... 38
### 2.6.3 Specification of induced velocities - $H$ function

### 2.6.4 Goldstein reduction factor background

### 2.6.5 Circulation distribution calculation using generalized Goldstein factors

### 2.6.6 Instability of calculated circulation distributions

### 3 DPLL Centerbody Inviscid Drag

#### 3.1 Centerbody representation

#### 3.2 Pressure forces on the body surface

#### 3.3 Lagally forces on body source rings

#### 3.4 Changes to the body drag force due to propulsor components

#### 3.5 Comparison of inviscid drag calculation methods

#### 3.6 Comparison of inviscid drag errors with expected total friction drag

### 4 DPLL Duct Inviscid Drag

#### 4.1 Duct Representation

#### 4.2 Pressure forces on the duct meanline

#### 4.3 Kutta-Joukowski forces on the duct meanline

##### 4.3.1 Method 1: Splining the control point velocities

##### 4.3.2 Method 2: Splining the discrete vortex strengths

#### 4.4 Comparison of duct drag calculation methods

##### 4.4.1 Kutta-Joukowski: splined velocities vs. splined vortex strengths

##### 4.4.2 Kutta-Joukowski vs. panel pressure summation

#### 4.5 Duct-body inviscid $C_D$ results vs. ship speed
5 DPLL Viscous Drag

5.1 Centerbody friction drag ........................................... 70
  5.1.1 Selection of a frictional drag coefficient ................. 70
  5.1.2 Calculation of body friction drag .............................. 71
5.2 Duct friction drag ............................................. 74
5.3 Appendage drag coefficient ....................................... 75
5.4 Overall drag coefficient for vehicle components ............. 76
5.5 Duct-body friction $C_D$ results vs. ship speed ............. 77
5.6 Duct-body friction $C_D$ results vs. duct circulation .......... 78

6 DPLL Duct Design Procedure ........................................ 81

6.1 A hypothetical duct design ...................................... 81
6.2 Position of governing blade-row tip .............................. 82
6.3 Initial duct conditions: Analysis ................................ 83
6.4 Construction of duct design matrices ............................ 83
  6.4.1 RHS column matrix ........................................... 84
  6.4.2 Duct-on-duct influence functions ............................ 85
  6.4.3 Specification of total duct loading .......................... 86
  6.4.4 Specification of finite leading edge load ................... 86
  6.4.5 Duct normal velocity due to duct angle of attack .......... 87
  6.4.6 Duct normal velocity due to duct camber .................... 88
6.5 Problem solution and iteration of duct design ................... 88
  6.5.1 Matrix solution ............................................... 88
6.5.2 Duct design iterations ........................................ 89
6.5.3 Duct radius adjustment ........................................ 89

7 Lifting Line Design Procedure .................................... 91

7.1 A hypothetical design of blade rows ......................... 91
7.1.1 Background ...................................................... 91
7.1.2 Identification of a governing blade-row tip ................ 93
7.1.3 Growing lifting lines .......................................... 93

7.2 Spacing lifting lines .............................................. 94
7.3 Trailing vortices .................................................. 95
7.4 Distribution of blade-rows induced velocities ............... 96

8 DPLL Calculation of Propeller-Induced Velocities ............ 98

8.1 Lifting line representation of a propeller blade .......... 98
8.2 Self-induced velocities .......................................... 99
8.3 Circumferential mean induced velocities .................... 99
8.4 Velocities induced by other blade-rows ..................... 100
8.5 Distribution of control points on the lifting line .......... 100

8.6 Control point location for velocities induced by

other blade-rows ....................................................... 101

8.7 Control point and lattice spacing for CMV calculations .... 102

8.8 Circumferential mean velocity induced by

a lifting line propeller .............................................. 104

8.9 Application of propulsor-induced velocities to

the next DPLL iteration ............................................. 106
8.9.1 Convergence of the induced velocity field ........................................... 106
8.9.2 Induced velocity storage grid ................................................................. 107

9 Calculation of Forces for a Lifting Line Propulsor ......................................................... 110

9.1 Background ........................................................................................................ 110
9.2 New force coefficient definitions ........................................................................... 112
  9.2.1 Advance coefficient ....................................................................................... 113
  9.2.2 Thrust coefficient ......................................................................................... 113
  9.2.3 Torque coefficient ......................................................................................... 114
  9.2.4 Power coefficient ......................................................................................... 115
9.3 Lifting line force calculations with the new force coefficients .................................... 116
  9.3.1 Thrust .......................................................................................................... 117
  9.3.2 Torque .......................................................................................................... 118
9.4 Viscous forces on a blade-row .................................................................................. 119
9.5 Propeller efficiency ............................................................................................... 119
  9.5.1 Traditional propeller efficiency ..................................................................... 119
  9.5.2 Comparison of propulsors based on power consumption ............................... 121

10 DPLL Program Overview .............................................................................................. 123

11 Design Applications of DPLL .................................................................................... 129

  11.1 Convergence of force results for discretization variations ................................. 130
  11.2 Single blade-row propulsor: Vehicle force balance vs. advance coefficient 134
  11.3 Single blade-row propulsor: Power coefficient vs. duct loading ..................... 135
11.4 Other possible design techniques with *DPLL* .......................... 136

12 *Future Work on DPLL* .......................... 145

12.1 Vehicle afterbody variations ............................................ 145

12.2 Duct representation .......................................................... 146

12.3 Improved viscous drag models ........................................... 146

12.4 Optimization link to the design process ................................ 147

12.5 Viscous wake model .......................................................... 148

13 *Incorporation of Analysis Capabilities into PBD-14/Design* 149

13.1 Background ........................................................................ 149

13.2 Induced velocity decomposition and nomenclature .................... 151

14 *Design Overview* ................................................................. 153

14.1 Blade representation ............................................................ 153

14.2 Design in an effective inflow ............................................... 156

14.3 Coupling *PBD-14* with RANS ................................................. 158

14.4 Effective inflow in *PBD-14/Design* ......................................... 159

15 *Analysis Background* ............................................................ 163

15.1 Horseshoe influence function theory ....................................... 163

15.2 Hub and duct images ............................................................ 164

15.3 Horseshoe influence function verification .................................. 165

16 *Solving for Blade Circulation* .................................................. 167

16.1 Horseshoe influence function matrix ....................................... 167
16.2 Solved circulation ................................................. 168

17 Verification of Analysis in an Effective Inflow 170

18 Coupling PBD-14 /Analysis with RANS 180

19 Verification of Coupled Design and Analysis 187

20 Comparison of Coupled

PBD-14 /Analysis

with Experimental Results 191

21 Future work on PBD-14 198

21.1 Blade thickness and viscous modeling .................. 198

21.2 Tip-gap flows ................................................. 198

22 Conclusions ....................................................... 200

22.1 Derivation of new non-dimensional coefficients ........ 200

22.2 The new design procedure .................................... 201

22.2.1 Preliminary design ....................................... 203

22.2.2 Blade design ............................................. 204

22.2.3 Blade analysis ........................................... 205

22.3 Summary ....................................................... 206

A Cubic Spline Representation of Propeller Induced Velocities 207
# List of Figures

1-1 Notional body with a slowly tapering afterbody ........................................ 21  
1-2 Notional body with a highly tapered afterbody ........................................ 23  
2-1 Spacing of body control points and source rings ........................................ 34  
2-2 Flowchart for the construction of the generalized Goldstein reduction factor for a given geometry of the lifting line and trailing vorticity and the current radial circulation distribution ........................................ 43  
2-3 Quadratic least-squares fit to a circulation distribution calculated from generalized Goldstein factors ........................................ 46  
3-1 *DPLL* body control points and source rings. ........................................ 47  
3-2 Local inviscid velocity at the body control points. ........................................ 49  
3-3 Calculated inviscid $C_P$ along a body. ........................................ 50  
3-4 Body inviscid $C_D$ vs number of body control points. ........................................ 53  
3-5 Cumulative inviscid pressure and Lagally drag along a body ........................................ 54  
4-1 *DPLL* duct control points and vortex rings ........................................ 58  
4-2 Kutta-Joukowski calculation of duct cumulative inviscid drag. ........................................ 63  
4-3 Duct inviscid $C_D$ vs number of duct control points. ........................................ 64
4-4 No-load duct design for duct-body vehicle in uniform inviscid inflow.

4-5 Body, duct, and total $C_D$ (friction plus inviscid forces) for unpropelled vehicle versus ship speed in uniform inflow. The profile of the flow over the vehicle has no viscous correction applied. The duct is specified to have zero net circulation. The vehicle is composed of only a body and a duct.

4-6 Body, duct, and total $C_D$ (inviscid forces only) for unpropelled vehicle versus duct circulation in uniform inflow. The profile of the flow over the vehicle has no viscous correction applied. The vehicle is composed of only a body and a duct.

5-1 Frustum representation of body panels

5-2 Body, duct, and total $C_D$ for unpropelled vehicle vs. ship speed. The flow over the body has no viscous correction to the profile. The duct is specified to have zero net circulation.

5-3 Body, duct, and total $C_D$ including friction forces for an unpropelled vehicle vs. duct circulation. The flow over the body has no viscous correction to the profile. The vehicle is composed of only a body and a duct.

5-4 Example of duct with extreme negative (clockwise) circulation to illustrate limits to the amount of load that a duct can be expected to carry in an effort to change overall powering of vehicle.

6-1 Duct design matrix for 5 vortex ring example

6-2 Duct solution column matrix for 5 vortex ring example
6-3 Duct RHS column matrix for 5 vortex ring example .................. 85
6-4 Normal vector to the duct meanline .................................. 85

8-1 Field point locations for circumferential mean velocity calculations due
to a constant pitch helical horseshoe vortex. ............................. 103
8-2 Plot of DPLL circumferential mean axial induced velocities due to a
horseshoe vortex with trailing vortices at r/R=0.4 and r/R=0.8. The
control point radius was varied along this segment from r/R=0.41 to
r/R=0.79. .................................................. 103
8-3 DPLL velocity grid points for storage of propulsor induced velocities 107

10-1 Flowchart of program flow within DPLL .............................. 124
10-2 Flowchart for the duct design process ................................. 125
10-3 Flowchart for the analysis of the body and duct ..................... 126
10-4 Flowchart for the propulsor design process .......................... 127
10-5 Flowchart for the force calculations ................................. 128

11-1 Force coefficients for a single blade-row vehicle in uniform inflow with
varying numbers of body control points/source rings. The duct is spec-
ified to carry no circulation. The inflow to the propulsor has no viscous
correction to its profile. ........................................ 131

11-2 Force coefficients for a single blade-row vehicle in uniform inflow with
varying numbers of duct control points/vortex rings. The duct is spec-
ified to carry a circulation of 0.01. The inflow to the propulsor has no
viscous correction to its profile. ........................................ 132
11-3 Force coefficients for a single blade-row vehicle in uniform inflow with varying numbers of vortex segments on the lifting line. The duct is specified to carry a circulation of 0.01. The inflow to the propulsor has no viscous correction to its profile. 133

11-4 Force coefficients for a single blade-row vehicle in uniform inflow with varying numbers of vortex segments used to model the trailing vortices. The duct is specified to carry a circulation of 0.01. The inflow to the propulsor has no viscous correction to its profile. 134

11-5 Single blade-row propulsor at varying advance coefficient. The duct carries a small but constant amount of circulation. The degree of self-propulsion of the vehicle is indicated by how closely the total force coefficient of the vehicle approaches zero for a given advance coefficient. The inflow to the propulsor has no viscous correction to its profile. 136

11-6 Variation in power coefficient for vehicle self-propulsion vs. a variation in duct circulation. The propulsor is a single blade-row propulsor on an afterbody with small taper. The inflow to the propulsor has no viscous correction to its profile. 137

11-7 Duct geometry for minimum self-propulsion power coefficient from Figure 11-6. The propulsor is a single blade-row and the afterbody can be seen to have small taper. 138

11-8 Geometry of a contra-rotating ducted propulsor on a shaft will small taper. 140

11-9 Geometry of a single blade-row propulsor with small duct-body clearance. 141
11-10 Axial induced velocity of the geometry shown in Figure 11-9. 142
11-11 Radial induced velocity of the geometry shown in Figure 11-9. 143
11-12 Tangential induced velocity of the geometry shown in Figure 11-9. 144

14-1 PBD-14 Blade B-spline net. 154
14-2 PBD-14 Evaluated Vortex Lattice Blade Surface. 155
14-3 Flowchart of PBD-14 /Design in a prescribed effective inflow. 158
14-4 Flowchart of PBD-14 /Design using RANS inflow iterations. 160
14-5 PBD-14 coupled design convergence of blade shape iterations. 162

16-1 Contour plot of PBD-14 /Design residual normal velocities on the blade. 169

17-1 Flowchart of PBD-14 /Analysis in a prescribed effective inflow. 171
17-2 Five blade sample propeller for design/analysis verification. 173
17-3 Design vs. analyzed circulation by radius, no hub or duct images. 173
17-4 Design vs. analyzed circulation over entire blade, no hub or duct images. 174
17-5 Four blade sample propeller for design/analysis verification. 174
17-6 Design vs. analyzed circulation by radius, hub images used. 175
17-7 Design vs. analyzed circulation over entire blade, hub images used. 175
17-8 Design vs. analyzed circulation by radius, hub and duct images used. 176
17-9 Design vs. analyzed circulation over entire blade, hub and duct images used. 176
17-10 Five blade sample propeller for design/analysis verification: tapered body. 178
17-11 Design vs. analyzed circulation by radius, tapered body, hub images used. .......................... 178

17-12 Design vs. analyzed circulation over entire blade, tapered body, hub images used. ................. 179

18-1 Flowchart for PBD-14 /Analysis Using RANS Inflow Iterations. ... 184

18-2 Convergence of propeller forces versus number of PBD-14 /RANS iterations for DTRC propeller 4577 with coupled PBD-14 /Analysis... 186

19-1 Propeller geometry for coupled design/analysis verification: straight shaft. .......................... 188

19-2 Radial distribution of circulation for coupled PBD-14 /Design vs. coupled PBD-14 /Analysis, no hub or duct images. .................... 189

19-3 Circulation over entire blade for coupled PBD-14 /Design vs. coupled PBD-14 /Analysis, no hub or duct images. .................... 190

20-1 Propeller 4119, used for coupled PBD-14 verification .................. 192

20-2 PBD-14 results without thickness for Propeller 4119 .................. 194

20-3 PSF-10 and experimental results for Propeller 4119 .................. 196

20-4 PSF-10 results with thickness variations for Propeller 4119 ........ 197

22-1 Schematic of the traditional propulsor design process ............... 202

22-2 Schematic of the new design process for propulsors on highly tapered vehicles. ....................... 202

A-1 Cubic spline representation of a discrete velocity jump .......... 208

A-2 Cubic spline extrapolation of a discrete velocity jump .......... 209
A-3 Cubic spline representation of a smooth velocity variation . . . . . . . 210
List of Tables

10.1 DPLL process description locations ........................................ 124

20.1 Geometry of DTRC Propeller 4119. ..................................... 193
20.2 Propeller 4119 forces from PBD-14 and experiment ................. 195
20.3 Propeller 4119 forces from PSF-10 and experiment ................. 195
20.4 Propeller 4119 inviscid forces from PSF-10 and PBD-14 .......... 196
Chapter 1

Introduction

There is currently great interest in the design of free swimming underwater vehicles for a variety of applications [2, 7]. These vehicles are often axisymmetric with afterbodies that have some small or moderate amount of taper. A notional profile of a body with a slowly tapering afterbody is shown with inviscid streamlines in Figure 1-1. The slowly tapering afterbody is chosen in an effort to streamline the body and delay or avert separation of the flow over the vehicle. Flow can separate from a body as the pressure increases towards the after end of the vehicle [55].

Very often a ducted propulsor is considered for some of these vehicles because a duct can help to provide a quiet, efficient propulsor that is protected from entanglement and impact [7]. The propulsors for these vehicles are often combinations of rotating and fixed blade-rows.

Hydrodynamic design of marine propellers is typically accomplished in three steps. The first step, preliminary design, is traditionally the calculation of optimum radial distributions of blade forces. The second step is the design of a blade surface which produces the optimum force distribution. The third step is an analysis of the designed
blade surface.

Existing blade design methods for marine propulsors are very often based upon preliminary lifting line design procedures which represent a propeller by a set of straight, radial lifting lines. These preliminary techniques (such as PLL-4.1, developed at MIT[9, 13]) represent each blade of the propeller with a lifting line and assume that the propulsor wake is purely helical, i.e., vortices which trail aft off the propulsor blades are convected downstream on constant radius cylinders. Helical vortices of constant radius allow the use of the very efficient asymptotic formulas developed by Wrench [56] (to compute the velocities induced by a particular blade-row of the propulsor on itself) and the formulas of Hough and Ordway [23] (for the calculation of the circumferential mean velocities induced on a blade-row of the propulsor by another blade-row). Clearly not all vehicles have propulsor designs with straight radial blades and a wake geometry that is highly cylindrical, but some geometries with small afterbody taper are close enough that such cylindrical methods can be used effectively.
The gently tapered vehicle afterbody form could be adjusted to produce a vehicle with more volume aft. The increased volume aft for a given maximum vehicle diameter and length necessarily means a less gently tapered afterbody. Consideration of propulsor designs with such increased conicity of the flow around the propulsor makes the constant diameter helical wake assumptions less appropriate. Likewise, the purely helical models in general assume a constant hub radius for all propulsors. This assumption is inappropriate for many tapered bodies. The notion of bodies with increased taper indicates the need for a new design methodology. This improved propulsor design technique may provide these vehicles with improved performance in range, speed, or maneuverability strictly as the result of propulsor designs that are tailored to operation with tapered afterbody shapes.

Submarine hull designs have been proposed that have a constant or very slowly changing diameter extending almost to the stern of the vehicle, at which point there would be a rapid diameter decrease to the after end [7]. The goal is to use the acceleration of the flow just forward of the propulsor to cancel the deceleration effects of the very highly tapered body in that region. This pairing could be used to prevent separated flow over the highly tapered stern which would have occurred in the absence of the propulsor. A notional representation of such a highly tapered vehicle was created by the author based on the description in [7] and is shown with inviscid streamlines in Figure 1-2.

A propulsor design technique accounting for high amounts of afterbody taper would allow designers to consider vehicle afterbody designs with more taper aft than is traditional. An alternate preliminary design technique which models contracted stream-
tube geometries is a streamline curvature method [47, 57]. This technique has been developed and applied over the years at the Applied Research Laboratory at Pennsylvania State University [1]. A shortcoming of streamline curvature methods, however, is that they typically do not model the flow on the outside of the duct. The duct streamline is used as the outer flow boundary for the method. Therefore the effect of the duct is not completely accounted for.

The work included here is oriented toward the improvement of propulsor designs for vehicles with high afterbody taper. This was accomplished with a new preliminary design technique and a new analysis technique.

The new preliminary design technique, DPLL, is a lifting line technique for the preliminary design of single or multiple blade-row ducted propulsors. The circulation distribution on each blade-row of the propulsor is solved from a designer input radial distribution of tangential induced velocities on that blade row. Input of the induced velocity distribution during preliminary design allows the designer some control over the induced velocity distribution in later steps of the design. DPLL also performs
a duct mean-camber design process to allow the designer to investigate the effect of variations in duct loading and shape on propulsor performance. Additionally, by changing the input afterbody shape, the effect of vehicle afterbody variation on propulsor performance can be examined.

Prior to the current work, analysis of tapered propulsor designs was typically carried out with separate analysis codes. These codes contained only rudimentary treatments of the contraction of the streamtube geometry and required time consuming conversions of the output files from the blade design program PBD-14/Design into the different format of the analysis codes.

In contrast, the new PBD-14/Analysis technique is integrated with the existing PBD-14/Design blade design program. Use of PBD-14 as either a design or analysis code is controlled with a single, common input file. All other input and output files are of identical format between design and analysis. Both the design and analysis techniques contain the logic for operating the propulsor in an input effective inflow. It is also possible to carry out a design or analysis in PBD-14 while coupled with an axisymmetric viscous flow solver. PBD-14/Design and PBD-14/Analysis are capable of modeling a propulsor and afterbody with high conicity. The resultant streamtube contraction is also accounted for in both the design and analysis methods.

Combined with the existing PBD-14/Design, the new preliminary design technique DPLL and the new analysis technique PBD-14/Analysis result in a new propulsor design process. This new design process allows the investigation of the propulsive characteristics of multiple blade-row ducted propulsors on highly tapered afterbodies.

It is possible that submarines more highly tapered aft than is traditional will result
in greater propulsive efficiencies. On the other hand, experience with such hull forms already in existence indicate that the efficiency gains over traditional designs are only marginal [19]. Therefore, the important benefit of underwater vehicles designed with highly tapered afterbodies will be the increased interior volume aft.

High taper aft on the vehicle would provide more space inside of the body in the region where most of the propulsion systems are located. The increased aft volume could allow the vehicle a greater energy storage capacity, payload volume and weight, or improved handling and station keeping characteristics. The change in the distribution of submerged volume may also allow designers additional freedom for improvements to the design of the remainder of the vehicle. An example of an improvement resulting from changes to the buoyancy distribution might be a fore-and-aft repositioning of some of the more massive components of the propulsion machinery. Effects of this and other improvements could include vehicle size, weight, length, and cost.
Chapter 2

A Lifting-Line Theory for Ducted Propulsors on Highly Tapered Afterbodies

2.1 Background

The design of propulsors for axisymmetric underwater vehicles typically begins with a preliminary examination of possible propulsor design conditions with a lifting line technique. An example is PLL-4.1 [9, 8], developed at the MIT Marine Hydromechanics Laboratory. The blades of the propulsor are each represented with a straight radial lifting line. Trailing vorticity is convected downstream along constant radius cylinders. The lifting line results and the selected design point parameters are then used as input to subsequent steps in the design process.
2.1.1 A need for generalized streamtube geometries

One major limitation of lifting-line techniques similar to PLL-4.1 is the representation of the trailing vortex wakes from a given propulsor component by constant radius and constant pitch helices. Though simple and effective for calculations on vehicles with little or no conicity to the flow in the proximity of the propulsor, this approximation is inappropriate for vehicles with highly tapered sterns. Computation of trailing vortex influence functions are more complicated and time consuming for an afterbody with generalized geometry.

An additional approximation of PLL-4.1 is a hub of constant radius for all of the blade-rows of the propulsor. The design of a propulsor in PBD-14 allows for hubs of varying radius from one blade-row to the next. Clearly, tapered designs carried out in PBD-14 based upon design parameters extracted from a preliminary design technique which ignores hub taper are expected to have shortcomings.

2.1.2 Improved propeller-induced velocity distributions

Blade design tools such as PBD-14/Design, developed at MIT, are already capable of representing a highly tapered afterbody and the resultant high conicity streamtube geometry. Likewise, PBD-14/Design can also represent multi-component propulsors and their ducts. The circulation input to PBD-14/Design, though, is currently obtained from PLL-4.1 or some designer-adjusted PLL-4.1 result. As stated above, there is a basic inconsistency with performing a design of a highly tapered propulsor with a design code such as PBD-14 using a circulation distribution or design conditions that were obtained with the assumption of cylindrical trailing vortex trajectories.
Circulation distributions that appear to be without large gradients may, more importantly, have induced velocity distributions on the blade of the propeller that are erratic. Computational experimentation with \textit{PBD-14} using different circulation distributions shows a sensitivity to induced velocities on the blade during the design process. In general, smoothly varying radial distributions of induced velocities lead to convergent and successful blade designs for propellers. On the other hand, circulation distributions that turn out to cause large gradients in the induced velocities along the blades often result in blades that appear mangled and bent. The design process in \textit{PBD-14} is unsuccessful in its attempt to satisfy the kinematic boundary condition of zero flow through the surface of the blade in the presence of erratic induced velocity variations.

2.2 Present technique overview

The work detailed here describes \textit{DPLL}, a ducted propulsor lifting line design program with the ability to represent trailing streamtube geometries that are highly conical. The technique represents each propulsor blade-row by lifting-lines composed of vortex segments which do not necessarily lie along a straight line nor are they necessarily oriented in the radial direction: They may have some degree of rake to them. The propulsor duct is represented by vortex rings while the vehicle centerbody is represented by a submerged source ring arrangement.

This technique allows the propulsor to have multiple blade-rows with varying absolute hub radius from one to the next. \textit{DPLL} also calculates trailing vortex induced velocity influence functions for trailing vorticity that follows the flow around vehicles
with varying afterbody and duct taper.

In order to simplify the modeling, the convection of the trailing vorticity is carried out using propeller-induced velocities which are circumferential means. The use of a circumferentially axisymmetric velocity field is an approximation, but one that is expected to have very little effect on the trajectory of the trailing vortices. The major differences between local induced velocities and circumferential mean induced velocities are usually confined to a region about 1 propeller diameter upstream and downstream of the propulsor. In this model, though, the propulsor is a ducted design which tends to lead to a highly axisymmetric character of the flow within the duct. As the area within the duct is also the location of the blade-rows, it is expected that the circumferential mean induced velocities will vary only slightly from the local non-axisymmetric velocities there. Therefore, DPLL was developed using time averaged velocities.

DPLL also takes as input a designer-specified distribution of tangential self-induced velocity on each blade-row. The form of the self-induced velocity is currently specified as a linear distribution of tangential velocity from the hub to the tip of each blade-row. It is possible that a higher order curve will turn out to be a more appropriate option for the form of the induced velocity distribution, but DPLL was developed with a linear distribution for initial simplicity. The actual circulation distribution along each blade-row is computed within the program from the geometry and the input induced velocity distribution. In this manner, the form of the self-induced tangential velocity is guaranteed and the designer is provided with a resultant output circulation distribution on each blade-row which is known to induce the input tangential self-
induced velocity distribution.

Overall, this design technique is intended for large parametric studies where basic accuracy and speed are the main goals. The primary inputs to this theory are a fixed centerbody geometry, the axial location and extent of the duct, the magnitude of duct load, the number of blade-rows, and the form of the self-induced tangential velocity on each one of the lifting-line blade-rows. The inviscid force on each component of the vehicle and the viscous contribution to the drag of each component of the vehicle is calculated. These force calculations are used by the designer to determine self-propulsion points of the vehicle. Self-propulsion of the vehicle occurs when the total (inviscid plus viscous) axial forces from the blade-rows, the duct, and the body sum to zero. The important results of a given design with this theory are the duct shape and orientation, the distribution of circulation on each of the lifting-line blade-rows, and a measure of the propulsor power consumption at self-propulsion.

2.3 Duct representation

The duct in *DPLL* is of zero thickness and is treated as a mean camber surface. This mean camber surface is represented hydrodynamically by vortex rings on the camber surface. The vortex rings, and their associated control points, are spaced in arclength along the mean camber line according to the theory of Lan [44]. Lan's theory allows for the conversion of discrete vortex ring strengths into sheet strengths (See Equation 4.2 and the associated text) for panels along the duct to aid in force calculations. Another use of the duct sheet strength is to allow the formulation of a design constraint on the duct shape which guarantees a finite load at the leading
edge of the duct.

The duct is input to DPLL as a set of fourth-order B-spline vertices that closely approximate an input form of camber distribution. A form of the camber distribution along the duct is also input to the theory, as well as the total duct circulation. The designed duct shape is then computed to give the desired total circulation on the duct while guaranteeing finite loading at the duct leading edge. The form of the camber distribution as a function of duct chordlength is maintained throughout the design process due to the multiplicative nature of B-splines. That is, for a given camber increment that needs to be added to the present duct shape, the B-spline vertices are all simply multiplied by the same factor which automatically results in a proper increase or decrease of duct camber, while maintaining the initial camber form.

The axial location of the duct leading edge and the duct trailing edge are taken as input to the design procedure. It was deemed reasonable to assume that the designer will have a clear idea of the axial extent and location of the duct based upon other vehicle considerations such as maneuvering characteristics or a need to protect the rotating portions of the propulsor.

In general, each time the DPLL duct design routines are called upon to carry out a duct design, the strength of the body singularities and the strength and location of the blade-rows will have changed somewhat from the last time the duct was designed. The duct solution is iterative in nature with the rest of the design procedure and therefore accounts for unchanging criteria such as specified duct loading and axial extent as well as changing quantities like the body ring strength and blade-row characteristics as they vary as part of the entire design iteration process.
2.4 Vehicle body representation

It is fair to expect that the designer will have either a dictated centerbody shape that must be used or an idea of the shape of body that should be used that is clearer than any preconceived notions that the designer may have about the exact shape of the propulsor and its duct. Therefore, the shape of the vehicle body is taken as a known input to DPLL. Using a program written to convert an X,Y curve into an equivalent B-spline representation, the given centerbody shape is represented by a relatively small number of B-spline vertices. The advantage of the B-spline representation is its compactness when compared to the number of X,Y pairs that would be necessary to adequately represent the smooth centerbody shape with discrete points.

The body shape is input to DPLL as a set of fourth order B-spline vertices. The use of inherently smooth B-splines makes it easy to represent smooth vehicles, including those with highly tapered afterbodies. Additionally, B-splines can represent a complex shape with relatively little input data (the vertices) while allowing an unambiguous complete interrogation of that shape without any approximations incurred by interpolation. The B-spline representation of the centerbody should also allow for future improvements to the code to handle interactive afterbody shape adjustments to improve propulsor powering characteristics. An excellent example of an interactive scheme for adjustments to the body shape already exists [28]. In this technique, the designer uses the computer mouse to move the body B-spline vertices on the screen and immediately see the resultant new body. Duct manipulation is also available in the same manner. The propulsive characteristics of an actuator disk propelling the new body and duct are subsequently calculated. The interactive portion of this
technique is, unfortunately, computer hardware dependent.

The hydrodynamic representation of the body is a submerged source ring solution to the body shape. This representation is used on the recommendation of Neely of David Taylor Model Basin based on his experience with his PIP program [52]. Though the use of a panel method could certainly have been used for the surface of the body, it was unnecessary to go into the program development and associated time expense for the body surface calculation. The current purpose of the body representation is to provide a realistic approximation of the flow over the body in order to examine trends in propulsor power consumption characteristics for different propulsor configurations and afterbody taper. Certainly, future versions of this program may be more concerned with accuracy of body forces or flow details. At that time, the adoption of a different body representation may be warranted. The results from the body force calculations using the source ring technique, however, is shown in Section 3.5 to be surprisingly accurate.

The source rings are submerged beneath the centerbody surface a distance proportional to the local spacing of control points on the body surface. The body control points are cosine spaced in arclength along the body surface starting at the nose of the vehicle and ending at the first blade-row. The surface space between blade rows, if multiple rows are present, is likewise cosine spaced. The space from the last blade-row to the tip of the stern is cosine spaced. This control point spacing scheme creates a concentration of control points around the hydrodynamically interesting portions of the body surface; that is, the bow, the stern, and the areas surrounding the junction of each propulsor lifting-line with the body.
2.5 Lifting line blade-rows

2.5.1 Geometry

The ducted propulsor in DPLL is idealized as up to three lifting-line blade-rows. A given blade-row can be specified to be either a stator or a rotor by the sign and magnitude of its input advance coefficient. The lifting lines representing the blades of each blade row are all broken up into the same number of lifting line segments along their lengths. The lifting line for the blade-row that is farthest upstream on the propulsor is initially represented with a continuous smooth line that is positioned so that it is everywhere very close to being normal to the local velocity vector. That continuous line is then broken up into a specified number of straight vortex segments of uniform length. This positioning and spacing of the lifting lines normal to the local flow vector is described in Sections 7.1.3 and 7.2.
2.5.2 Advance coefficient

Typically, the operating point of a propeller is described by a non-dimensional advance coefficient, \( J \). Both \( J_A \) and \( J_S \) are used.

\[
J_A = \frac{V_A}{(n \times D)} \tag{2.1}
\]

where

\( V_A \) = velocity of advance of propeller
\( n \) = propeller rotations per second
\( D \) = propeller diameter

\[
J_S = \frac{V_S}{(n \times D)} \tag{2.2}
\]

where

\( V_S \) = ship speed
\( n \) = propeller rotations per second
\( D \) = propeller diameter

Both \( J_A \) and \( J_S \) are non-dimensionalized using propeller diameter. For an open propeller, with very small or zero tip chord, on a cylindrical hub, these definitions are fairly unambiguous. In ducted cases, the tip chord often is no longer very small and it roughly follows the duct inner surface which is not necessarily of constant radius. Some confusion can arise about which radius (tip leading edge, tip trailing edge, or tip mid-chord) should be used to define the propeller. The problem becomes worse...
when the hub radius varies along the chord of the blade, and in the case of multiple blade-rows, also from row to row.

A radius difficulty specific to DPLL is that blade-rows other than the governing blade-row have radii which vary as the duct shape varies during the duct design process and from one iteration to the next. Therefore, these blade-rows have changing radii throughout a run of the program. Thus, an advance coefficient was defined that was instead dependent upon the maximum radius of the body and the ship speed. Typically this maximum body radius, $R_B$, is the radius along the parallel mid-body of the vehicle. This new advance coefficient, $J_B$, is

$$J_B = \frac{V_S}{(n \times R_B)} \quad (2.3)$$

where

$V_S = \text{ship speed}$

$n = \text{propeller rotations per second}$

$R_B = \text{maximum radius of the body}.$

The sign of the advance coefficient, positive or negative, indicates the direction of rotation of each blade-row. A positive advance coefficient corresponds to a right-handed turning propeller and conversely for a negative advance coefficient. For a stator, the input advance coefficient is some very large number to indicate to the program that the blade-row is essentially non-rotating.

Detailed in Section 2.6 is the process by which the circulation strengths of all of the vortex segments making up the propulsor geometry are assigned. The circulation distribution on each blade-row is calculated using designer input distributions of
tangential velocities induced on each lifting line by its own circulation and trailing vorticity.

2.6 Radial distribution of circulation and induced velocities

2.6.1 Representation of circulation on the lifting line

The continuous radial distribution of circulation, $\Gamma$, along each lifting-line blade-row is represented by

$$\Gamma(r) = \frac{2 \pi r}{Z} \cdot \frac{2 \bar{u}_T}{2}$$

(2.4)

where

$$\bar{u}_T = \text{circumferential mean tangential induced velocity on the lifting line}$$

$$Z = \text{number of blades on the propeller}$$

The common non-dimensional form of the circulation, $\Gamma$, is $G$, and utilizes the propeller radius:

$$G(r) = \frac{\Gamma(r)}{2 \pi R V_S}$$

(2.5)

where

$$R = \text{propeller radius}$$

$$V_S = \text{ship speed}.$$
In the current work, though, different blade-rows can have different radii and those radii in turn can vary throughout the design procedure as the duct design is adjusted. The use of propeller radius for non-dimensionalization of circulation was therefore deemed dangerous and a new non-dimensionalization was developed using body maximum radius, as with the advance coefficient.

\[ G(r) = \frac{\Gamma(r)}{2\pi R_B V_S} \]  

(2.6)

where

\[ R_B = \text{maximum radius of the body} \]

\[ V_S = \text{ship speed}. \]

Substituting Equation 2.4 into Equation 2.6 gives the complete form of the non-dimensional circulation on the lifting line.

\[ G(r) = \frac{2\pi r \cdot 2 \bar{u}_T}{2\pi R_B V_S Z} = \frac{2 \bar{u}_T}{V_S R_B} \cdot \frac{r}{Z} \]  

(2.7)

In lifting-line theory, this continuous circulation distribution is instead replaced by a finite number of vortex segments with constant circulation strength along each segment. The circulation strength is allowed to vary from one vortex segment to the next.

2.6.2 Design circulation distributions

Traditionally, a propeller design is carried out according to some given distribution of circulation along the radius of a blade. This circulation distribution typically
comes from the solution to some preliminary scheme which optimizes the propeller thrust to torque ratio, such as the solution performed by MIT PLL-4.1 as developed by Coney [9]. However, the optimum circulation distribution for multiple blade-row ducted propellers is in general a constant along the radius.

The distributions obtained also often contain a bit of “art” as well when designers adjust the circulation at points along the radius in an effort to solve problems or meet some criterion. Sometimes the adjustments are carried out in an effort to have one blade-row cancel swirl introduced by another upstream. Blade-row tips or hubs are sometimes unloaded due to flow separation or cavitation problems. Often, the adjustments to the circulation distribution are to improve the distribution of velocities induced by the circulation. The adjustments, or the original “optimized” distribution of circulation itself, can lead to unexpected severe gradients in the induced velocity field.

Blade shape design, such as the vortex-lattice lifting-surface method in PBD-14, is carried out in the presence of the total velocity field, which contains the induced velocity due to the propeller’s circulation. The flow field without the induced velocities is typically smooth in both the axial and the radial directions. Adding a distribution of induced velocity with severe gradients in the radial direction to the flow field demands a final blade design which is either impossible for design programs to create, or impossible or unreasonable to build if the design is completed. Such difficulties in PBD-14/Design, resulting from unfairness of the radial distribution of velocities, has been experienced recently [32]. It is assumed that a circulation distribution that induces a fair radial distribution of velocities will likewise induce a distribution of
axial velocities that will be smooth over the chordlength of the blades.

Clearly, a fair distribution of induced velocities is desired for increased chances of a successful final design. Therefore, a design scheme that produces such a smooth distribution at the preliminary level of the design is advantageous. The smooth distribution at the preliminary level, such as a lifting-line preliminary design, allows the designer to proceed to later steps in the design process with some level of confidence in the smoothness of the induced velocities.

2.6.3 Specification of induced velocities - $H$ function

In the present work, a function, $H$, which describes propeller's self-induced velocities as a function of radius, is defined as

$$H(r) = \frac{2 \frac{u_T^2}{V_S}}{R_B} \frac{r}{Z}$$

(2.8)

where

$R_B = \text{maximum radius of the body}$

$V_S = \text{ship speed}$

$u_T^* = \text{tangential induced local velocity at the lifting line}$.

$H(r)$ is the same as $G(r)$ except that it is based upon the local $u_T^*$ at the lifting line rather than the circumferential mean $\bar{u}_T$. The idea of the function $H(r)$ was developed by Kerwin [40]. His original formulation was the solution of the radial distribution of circulation on a single blade-row radial lifting-line propeller. The process was based upon an input linear distribution of $H(r)$ from which the circulation, $G$ (Equation 2.5), was solved, together with the induced velocities and propeller forces.
Solution of the discrete circulation strengths radially along the blades was made possible with the construction of influence functions which describe the local velocities which would be induced at a given control point along the blade by the presence of each spanwise vortex and its trailing vortices, assuming they were of unit circulation strength. The circulation distribution was solved for iteratively while the the trailing vortex pitch was adjusted iteratively to match the total flow at the lifting-line during each step.

2.6.4 Goldstein reduction factor background

As the number of blades on a propeller is increased, the circumferential mean tangential induced velocity from \([G(r)]\) approaches the local velocity from \([H(r)]\) on the lifting line. But, in general, the circumferential mean velocity is different from the local velocity on the blades for a finite number of blades. The relationship between \(\bar{u}_T\) and \(u^*_T\) was solved by Goldstein in 1929 [21] for the special case of an optimum radial distribution of circulation in uniform flow. This relationship is called the Goldstein reduction factor, \(\kappa\), where

\[ \kappa(r) = \frac{u^*_T(r)}{\bar{u}_T(r)}. \] (2.9)

The Goldstein reduction factor for a particular propeller depends only on the geometry of the problem [37] (number of blades, geometry of the trailing vorticity, and the radius). Thus, the construction of the Goldstein reduction factor for a particular geometry results in a relationship between the specified induced velocity function, \(H(r)\), and the circulation, \(G(r)\), for a single blade-row propeller.
2.6.5 Circulation distribution calculation using generalized Goldstein factors

Goldstein's work was for a special case, as discussed above. The present use of a Goldstein factor should therefore be called a "generalized" Goldstein factor. Instead of results for an optimum case only, the Goldstein factor here is recalcultated during each iteration for the current blade-row geometries and circulation distributions.

The strengths of the discrete spanwise vortex segments on each blade-row of the propulsor are calculated using the known $H(r)$ function for that row and a calculated radial distribution of the generalized Goldstein factor, $\kappa(r)$. The distribution of $H$ determines the local self-induced tangential velocities while the Goldstein factor relates the local self-induced velocities to the self-induced circumferential mean velocities.

The calculation of $\kappa$ is carried out for the geometry of the spanwise and trailing vortex system of the blade-row in question, along with the most recently calculated circulation distribution on that blade-row. Though it seems circular to require that one know the circulation distribution before it can be calculated, this is an iterative procedure where a guess at the circulation distribution allows calculation of the generalized Goldstein factor distribution along the blade and hence the circulation.

The lifting line representation of a blade in this work includes blade rake and a centerbody of arbitrary and possibly varying diameter in the area of the propulsor. It is assumed that the ratio of local tangential velocities to circumferential mean tangential velocities for the actual geometry and loading distribution is approximated by the same ratio for a straight radial lifting line with a similar radial distribution of loading. This approximation involves a constant-radius cylindrical hub geometry.
Fig. 2-2: Flowchart for the construction of the generalized Goldstein reduction factor for a given geometry of the lifting line and trailing vorticity and the current radial circulation distribution

and constant radius trailing helical vortices.

The local velocity induced at each control point on the straight radial lifting line is calculated by summing the induced velocity effect of each spanwise vortex segment on the line, together with the induced velocities of the trailing vortex helices from each segment. The local induced velocities are calculated from the formulas of Wrench [56], the strength of each vortex segment, and the total local pitch angle, $\beta_i$ [37].

The influence on the blade-row of the hub and the duct in the real geometry is approximated by including hub and duct images. As in PLL-4.1, the strength of the spanwise image vortex and its trailers is equal to the strength of the spanwise vortex which is being imaged. The radius of the image trailing vortices is assigned as in
PLL-4.1 [8]:

\[ r_2 = \frac{r_o^2}{r_1} \]  \hspace{1cm} (2.10)

where \( r_1 \) is the radius of the blade trailing vortex to be imaged, \( r_2 \) is the radius of the image trailing vortex, and \( r_o \) is the radius of the solid boundary (i.e. the duct or hub surface). For example, in the case of a hub image, \( r_o \) is the hub radius. For an image in the duct, \( r_o \) is the duct surface radius.

The hydrodynamic pitch of all image vortices in the hub is set to the hydrodynamic pitch of the blade at the lifting line segment immediately next to the hub. Likewise, the pitch of all of the duct image trailing vortices is set to the pitch of the blade at the blade tip vortex segment.

The circumferential mean tangential induced velocities at each control point on the lifting line are calculated as the summed induced effect of all of the lifting line segments. Calculation of the circumferential mean induced tangential velocities is performed with the formulas of Hough and Ordway [23] and the local hydrodynamic pitch angle, \( \beta_i \).

The summed local induced tangential velocity and the summed circumferential mean induced tangential velocity are presently known at all of the control points along the radius of the straight lifting line approximation to the real geometry. The generalized Goldstein reduction factor is easily calculated from Equation 2.9. Knowledge of the \( H \) function, \( H_i \), plus the generalized Goldstein factor, \( \kappa_i \), at each control point allows calculation of the new circulation at each control point, \( \Gamma_i \).

\[ \Gamma_i = \kappa_i \times H_i \]  \hspace{1cm} (2.11)
The new circulation strengths in turn allow the program to create a new vortex geometry from which the generalized Goldstein factors can again be calculated. The entire generalized Goldstein calculation method is shown schematically in Figure 2-2.

2.6.6 Instability of calculated circulation distributions

Generalized Goldstein reduction factors for a blade-row are calculated from the most recent radial distribution of circulation and the blade-row geometry. The resultant Goldstein factors are subsequently used to recalculate the circulation distribution. Experimentation with this technique showed that small computational errors in the calculation of the Goldstein factors were passed on to the calculated circulation distribution and hence to the subsequent re-calculation of the Goldstein factors. For this reason, the Goldstein factor scheme described in the previous section is unstable and resulted in divergent circulation distributions.

The generalized Goldstein factor calculation of circulations was made to be convergent with a least-squares best-fit line to the calculated circulation distribution for each blade-row (Figure 2-3. In this way, a fair distribution of circulation can be used to calculate the next Goldstein factors and computational errors are not passed on to future iterations. At present, the least-squares fit is a quadratic curve. It is possible that improvements to the technique could be realized from a different least-squares fit to the circulation.
Figure 2-3: Quadratic least-squares fit to a circulation distribution calculated from generalized Goldstein factors.
Chapter 3

DPLL Centerbody Inviscid Drag

3.1 Centerbody representation

As detailed more fully in Section 2.4, the vehicle centerbody in DPLL is represented hydrodynamically by control points on the body surface and an equal number of source rings submerged within the body along normals to the body surface extending in from each control point. The kinematic boundary condition of zero flow normal to the vehicle centerbody is satisfied at the control points of the axisymmetric body. These control points are cosine spaced in arclength along the body as in Figure 3-1.

Figure 3-1: DPLL body control points and source rings.
Potential flow theory dictates that a closed body in a steady inviscid flow will be subjected to zero net force from that flow. The local force at any point on or within the body in the present problem can be broken up into a force perpendicular to the body centerline (side force) and a force parallel to the body centerline (drag force). Due to the form of DPLL with an axisymmetric vehicle, propulsor, and duct undergoing forward motion with zero yaw angle, the side forces will always be axisymmetric and sum to zero both locally and for the entire body. This will be true for both inviscid and viscous flow.

Drag force on a bare body will likewise sum to zero in inviscid flow. But with the addition of viscous forces and pressure forces due to the presence of a duct and propulsor, the drag no longer necessarily sums to zero. Thus, forces on the body in the axial direction (drag) are the important forces for this problem.

The comparison below of different drag calculations was carried out for a bare body in inviscid uniform flow. This allowed comparisons of the techniques based on how close to zero each summed the net body drag force.

### 3.2 Pressure forces on the body surface

The local velocity, \( q \), at each one of the control points on the body surface can be calculated by summing the induced velocity of the body source rings and adding the inflow velocity from ship speed, \( V_S \). The kinematic boundary condition was satisfied in the solution to the problem, and therefore the flow at each one of the control points is tangent to the body there as shown in Figure 3-2.

From this local velocity, a coefficient of pressure, \( C_P \), can be calculated at each
Figure 3-2: Local inviscid velocity at the body control points.

control point:

\[ C_P = \frac{p - p_\infty}{\frac{1}{2} \rho V_S^2} \]  
(3.1)

where

\[ p - p_\infty = \frac{\rho}{2} (q^2 - V_S^2) \]  
(3.2)

such that

\[ C_P = 1 - \frac{q^2}{V_S^2} \]  
(3.3)

Figure 3-3 shows the calculated \( C_P \) over the length of a body using 100 control points.

The \( C_P \) expression dictates that \( u = 0 \) and \( C_P = 1 \) at a stagnation point. In undisturbed flow, \( u = V_S \) and \( C_P = 0 \).

The pressure drag, \( D \), could be calculated by an integral over the length of the body

\[ D = \int_0^L p \ A \ dL \]  
(3.4)
where

\[ p = C_P \frac{1}{2} \rho V_S^2 \]  

(3.5)

and \( A \) is some representation of the axial projected area of the body.

In this problem the body surface velocity is a discrete value at each control point. The body surface can be "paneled" by associating the area local to each control point with the velocity calculated at that control point. \textit{DPLL} calculates panel boundaries as the midpoints, in body surface arclength, between adjacent control points.

The pressure calculated at each control point can be treated as a constant pressure acting over the panel surrounding that control point. Thus, each pressure, multiplied by the axial projected area of its panel, can be used to sum the pressure drag force
over \( n \) panels on the body:

\[
\text{Drag}_p = \sum_{i=1}^{n} C_p, \frac{1}{2} \rho V_s^2 \pi (r_{2,i}^2 - r_{1,i}^2),
\]

(3.6)

where \( r_1 \), and \( r_2 \), are the radii of the front and back edges, respectively, of the \( i \)th panel.

This dimensional drag force is converted to a coefficient of drag from pressure,

\[
C_{D_p} = \frac{\text{Drag}_p}{\frac{1}{2} \rho V_s^2 \pi R_B^2},
\]

(3.7)

where \( R_B \) = maximum radius of the vehicle body.

### 3.3 Lagally forces on body source rings

The Lagally force per unit length on a two dimensional point source, due to an onset flow, is

\[
\text{Force} = \rho u \sigma^{2D}
\]

(3.8)

where \( u \) is the local axial velocity at the point source without the source induced velocities and \( \sigma \) is the point source strength in units of \( \frac{\text{length}^2}{\text{time}} \). Due to axisymmetry, this expression can be extended to the force on a source ring of radius \( R \) simply by multiplying this two-dimensional point source force by the length of the source ring, \( 2\pi R \), creating an expression with the source ring strength in units of \( \frac{\text{length}^3}{\text{time}} \) and the result in units of force.

Calculation of the axial Lagally force on a given source ring is based on the local axial velocity at that ring. To obtain the local axial velocity, the axial velocities
induced by each of the body source rings, except the ring whose force is in question, are added to the uniform axial inflow velocity.

The force on the body, where the body is represented hydrodynamically by \( n \) source rings, can now be calculated as the summation of the forces on all of the source rings:

\[
Drag_L = \sum_{i=1}^{n} \rho u_i \sigma_i^{3D},
\]  

(3.9)

where \( u_i \) and \( \sigma_i^{3D} \) are the source ring local velocity and strength, respectively.

Like the pressure drag, this dimensional Lagally drag force can be converted to a coefficient of drag due to Lagally forces, \( C_{DL} \):

\[
C_{DL} = \frac{Drag_L}{\frac{1}{2} \rho V_s^2 \pi R_B^2}
\]  

(3.10)

where \( R_B \) is the maximum radius of the vehicle body.

### 3.4 Changes to the body drag force due to propulsor components

For practical use, the drag calculation using either of the above techniques must be adjusted when the bare body is replaced by a body with components such as a duct and/or propulsor. For both the panel pressure summation and the Lagally's Theorem techniques, local velocities necessary for the calculations need to include the induced velocities of the duct and/or propulsor, if present. Additionally, the solution of the strengths of the source rings representing the body will be different when solved in the presence of the other components than it would have been if treated as a bare
3.5 Comparison of inviscid drag calculation methods

As stated above, the net drag on a bare body in potential flow should be zero. Figure 3-4 shows a comparison of a bare body drag coefficient calculated using both the panel pressure summation (Section 3.2) and Lagally's theorem (Section 3.3) for different numbers of control points on the body.

![Graph showing body inviscid $C_D$ vs number of body control points.](image)

Figure 3-4: Body inviscid $C_D$ vs number of body control points.

Clearly, the calculation for both techniques approaches zero with increasing num-
ber of control points. The technique based on Lagally's theorem, though, tends to approach zero net drag more quickly.

Figure 3-5 shows the cumulative drag along the length of a body using both panel pressure summation and Lagally theorem forces. The results shown are for a bare body with 100 control points in uniform inflow.

Figure 3-5: Cumulative inviscid pressure and Lagally drag along a body

The net drag forces shown for both techniques match those shown for 100 control points in Figure 3-4. The trajectory of the curve for the cumulative drag is very different for the two different techniques, but both sum very close to zero by the time the tail of the body is reached. The large difference in trajectory between the two methods is not a problem, though, as the shape of the Lagally summation has no physical significance.
3.6 Comparison of inviscid drag errors with expected total friction drag

Though neither inviscid technique sums exactly to zero, it can be shown that these errors are insignificant relative to the friction drag that can be expected from these types of bodies. In general, the resistance of these vehicles is dominated by frictional resistance [7]. A rough estimate of a typical coefficient of friction, based on wetted surface area $S$ and the ITTC friction coefficient [50], for streamlined bodies is

$$ C_F = \frac{D}{\frac{1}{2} \rho S V_s^2} = 0.003. \quad (3.11) $$

Typically, the form drag of such bodies can be expected to be a small percentage (around 10%) of the total drag, so the total viscous drag coefficient can be approximated with that from the friction estimate so that $C_D = 0.003$ normalized on wetted surface area. When we adjust this $C_D$ so that it matches the non-dimensionalization in the current work based on maximum cross-sectional area of the body, we have

$$ C_D = \frac{D}{\frac{1}{2} \rho V_s^2 A_B} = 0.057 \quad (3.12) $$

A comparison between the residual inviscid drag coefficients calculated in Figure 3-4 and the estimate of $C_D = 0.057$ helps to show how much error will be introduced from inviscid drag calculations which do not sum exactly to zero. The panel pressure summation has a residual $C_D$ of about 0.0006 while the Lagally theorem calculation has a residual $C_D$ of less than 0.0002. These errors correspond to about 1.0% and 0.4%, respectively, of the estimated total drag coefficient. The Lagally theorem
method (0.4%) was selected as the more accurate of the two drag calculations for the present formulations.
Chapter 4

DPLL Duct Inviscid Drag

4.1 Duct Representation

As detailed more fully in Section 2.3, the duct in DPLL is represented hydrodynamically by an equal number of control points and vortex rings located on the duct mean camber line. Both the control points and the vortex rings are cosine spaced in arclength along the duct camber line as shown in Figure 4-1. The kinematic boundary condition of zero flow normal to the mean camber surface of the duct is satisfied at the duct control points.

As with the body force calculations (see Appendix 3), the duct forces in the radial (side force) direction will always sum to zero and are therefore of no interest. The forces on the duct in the axial direction (drag) are of interest.

Net drag on a duct operating by itself in inviscid uniform flow should be zero. The comparison below of different drag calculation techniques is therefore carried out for a duct operating alone in uniform inviscid flow. Comparison of different calculation techniques is therefore based on how close to zero each one sums the net duct drag.
4.2 Pressure forces on the duct meanline

The calculation of a pressure coefficient, $C_P$ (see Equation 3.3), for the pressure on the duct meanline at each control point is based on a difference in velocity between the upper and lower surfaces of the duct at that control point. The uniform inflow and induced velocities from the other duct vortex rings in the problem result only in a velocity $u$ at the control point that varies smoothly from one side of the duct to the other — there is no jump in velocity due to inflow or induced velocities as you pass across the meanline. The discrete jump in velocity which accounts for the pressure difference across the duct comes from passing through the local vortex sheet which lies on the duct meanline.

In the same manner as the body is "paneled", the duct surface local to each control point and vortex pair can be associated with that control point by defining panel boundaries to lie at the midpoint, in duct arclength, of the distance between
adjacent duct control points. The strength of the discrete vortex, \( \Gamma_i \), located in the panel that extends from duct arclength \( s_{i-1} \) to \( s_i \) can be represented as

\[
\Gamma_i = \int_{s_{i-1}}^{s_i} \gamma(s) \, ds
\]  

(4.1)

where vortex sheet strength \( \gamma(s)_i \) varies along the length of the panel. In the case of DPLL, however, \( \gamma(s)_i \) is a constant over each panel, \( \gamma_i \), so that

\[
\gamma_i = \frac{\Gamma_i}{s_2 - s_1}.
\]  

(4.2)

From [37], a sheet strength of \( \gamma \) represents a jump in tangential velocity across the foil (or the duct, in this case) of magnitude \( \gamma \). A simplified example [50] is a linearized foil, along the x-axis, with constant vortex sheet strength \( \gamma_{constant} \), where the resultant x-velocity jump for the upper(+) and lower(-) sides is

\[
u_x^\pm = \mp \frac{1}{2} \gamma_{constant}.
\]  

(4.3)

Likewise, the velocity jump magnitude across the duct for a particular panel, \( i \), is \( 2u_d \) where

\[
u_d = \frac{1}{2} \gamma_i
\]  

(4.4)

So, the velocity on the upper side of the duct is \( u + u_d \) while the lower side has velocity \( u - u_d \). \( C_P \) on the upper and lower surfaces of the duct is

\[
C_P^{upper} = 1 - \frac{(u + u_d)^2}{U_{\infty}^2}
\]  

(4.5)
and
\[ C_{P}^{\text{lower}} = 1 - \frac{(u - u_d)^2}{U_\infty^2}. \] (4.6)

The difference in \( C_P \) across the duct, \( \Delta C_P \), is
\[ \Delta C_P = C_P^{\text{upper}} - C_P^{\text{lower}}. \] (4.7)

Substituting Equations 4.5 and 4.6,
\[ \Delta C_P = 1 - \frac{(u + u_d)^2}{U_\infty^2} - \left(1 - \frac{(u - u_d)^2}{U_\infty^2}\right) \] (4.8)

which results in
\[ \Delta C_P = -4 \cdot u \cdot u_d. \] (4.9)

We now have \( \Delta C_P \) from which the pressure on each panel can be calculated as
\[ p = \Delta C_P \frac{1}{2} \rho U_\infty^2. \] (4.10)

This calculated pressure for each panel, multiplied by the axial projected area of that panel, can be used to sum the pressure drag force discretely over \( n \) panels on the duct:
\[ Drag_P = \sum_{i=1}^{n} \Delta C_{Pi} \frac{1}{2} \rho U_\infty^2 \pi(r_2^2 - r_1^2) \] (4.11)

where \( r_1 \) and \( r_2 \) are the radii of the front and back edges, respectively, of each panel.

This dimensional drag force is converted to a coefficient of drag from pressure, \( C_{DP} \):
\[ C_{DP} = \frac{Drag_P}{\frac{1}{2} \rho U_\infty^2 \pi R_{\text{max}}^2} \] (4.12)
where $R_{\text{max}}$ = maximum radius of the vehicle body.

4.3 Kutta-Joukowski forces on the duct meanline

The Kutta-Joukowski theorem,

\[ \text{Lift} = \rho u \Gamma, \]

(4.13)

gives the lift force on a discrete vortex of strength $\Gamma$, which acts in a direction normal to the inflow velocity $u$. The axial force on the duct can be calculated in this manner by using the discrete strength of each vortex ring on the duct together with the radial velocity at that ring.

In DPLL, the discrete vortex ring strengths are known at the vortex ring locations while the velocity on the duct is known and well behaved only at the control point positions (see Figure 4-1). Clearly, Equation 4.13 holds only for a velocity and a vortex at the same location.

4.3.1 Method 1: Splining the control point velocities

The kinematic boundary condition on the duct is imposed at the control points. The velocity field resultant from the solution to the problem is therefore only well behaved at those control points. An array of cubic coefficients can be constructed for the radial velocity at the control points as a function of arc length along the duct. At the arclength location of the vortex rings, the splined radial velocity can then be evaluated. The resultant radial velocity at the location of the known strength vortex
ring can then be used to carry out a summation of axial Kutta-Joukowski forces on the duct.

4.3.2 Method 2: Splining the discrete vortex strengths

From the solution to DPLL, the discrete strengths of the duct vortex rings are known. An array of cubic coefficients can be constructed for vortex ring strength as a function of arclength along the duct. At the arclength location of the duct control points, the splined discrete vortex strength can then be evaluated. The resultant vortex strength at the control points and the radial component of the velocities there can then be used to carry out a summation of axial Kutta-Joukowski forces on the duct and a calculation of a duct drag coefficient.

4.4 Comparison of duct drag calculation methods

As stated above, the net drag on a duct operating alone in potential flow should be zero. Likewise, the drag coefficient,

\[ C_{D_P} = \frac{\text{Drag}}{\frac{1}{2} \rho U_{\infty}^2 \pi R_{\text{max}}^2}, \]  

(4.14)

where \( R_{\text{max}} \) = maximum radius of the vehicle body, should go to zero as well.

4.4.1 Kutta-Joukowski: splined velocities vs. splined vortex strengths

Figure 4-2 shows the cumulative drag calculated over the length of the duct using both of the Kutta-Joukowski methods above for 55 duct vortex rings. Clearly, the two drag calculation techniques are essentially identical both quantitatively and qualitatively.
Either method gives net duct drag values very close to zero, but Method 1 using velocities splined to the vortex rings consistently calculates a net drag closer to zero.

![Graph](image)

Figure 4-2: Kutta-Joukowski calculation of duct cumulative inviscid drag.

### 4.4.2 Kutta-Joukowski vs. panel pressure summation

Method 1 (which calculates Kutta-Joukowski forces with splined velocities) is used in Figure 4-3 to compare $C_D$ calculated from Kutta-Joukowski with $C_D$ based on the panel pressure summation of the duct drag. The figure shows the drag coefficient calculated using both methods for various numbers of duct control points.

The inviscid duct $C_D$ shown in Figure 4-3 as calculated by a Kutta-Joukowski method approaches zero much more quickly than that from a summation of panel...
pressures. The Kutta-Joukowski calculation is carried out by method 1 as described in Section 4.3.1. A representative $C_D$ from [50] for an airfoil section at low angle of attack like the duct shape in this example, is about 0.007. Converting this to a $C_D$ which is normalized in the manner of the present work, $C_D = 0.011$. From Figure 4-3, the inviscid $C_D$ calculated by panel pressure summation is around -0.0002 while that from the method 1 Kutta-Joukowski is around 0.00001. These absolute errors in $C_D$ calculation by the two methods correspond to about -1.8% and 0.09%, respectively, of the representative total airfoil $C_D$. 

Figure 4-3: Duct inviscid $C_D$ vs number of duct control points.
4.5 Duct-body inviscid $C_D$ results vs. ship speed

As stated in the Sections 3 and 4, the inviscid force calculation for a component of the vehicle operating by itself in uniform flow should sum to zero. For the examples shown in those sections, that is the case. If the duct and the centerbody are now combined as one unit, together they can be considered as a single component. If the duct and body combination vehicle is then subjected to uniform inflow, the net inviscid drag should also sum to zero. The drag on the body or on the duct will not necessarily be zero, as they each are now operating in flow altered by the other. Together, the inviscid drag should vanish.

In the case of a duct and body vehicle where the duct has a prescribed circulation of zero, a short duct will be essentially aligned everywhere with the local flow over the body (See Figure 4-4). In this case, the duct surface will approximate a streamline of the flow. If the duct lies along a streamline of the body-only flow, as in Figure 4-4, the body will not “know” about the presence of the duct. Therefore, the body singularity strengths will change only negligibly from the body-only case and leave the body experiencing only uniform axial inflow. From before, we know that the body in uniform inflow should have zero inviscid drag. As the duct was specified to have zero load, the sum of body and duct inviscid force should again be zero.

4.6 Duct-body inviscid $C_D$ results vs. duct circulation

As the circulation of the duct is increased or decreased the flow around both the body and the duct is affected by the presence of the other component. Increased circulation on the duct is counterclockwise in this formulation and will cause increased flow speed
through the gap between body and duct. The increased flow speed results in decreased pressure there and a resultant increase in body inviscid drag. The positive circulation on the duct will likewise cause forward thrust on the duct. As before, considering the two components as one vehicle unit, the net inviscid force on the vehicle should still be zero. Figure 4-6 shows the coefficient of drag of the body, duct, and the entire vehicle for varying amounts of duct circulation. The inviscid sum is zero.
Figure 4-5: Body, duct, and total $C_D$ (friction plus inviscid forces) for unpropelled vehicle versus ship speed in uniform inflow. The profile of the flow over the vehicle has no viscous correction applied. The duct is specified to have zero net circulation. The vehicle is composed of only a body and a duct.
Figure 4-6: Body, duct, and total $C_D$ (inviscid forces only) for unpropelled vehicle versus duct circulation in uniform inflow. The profile of the flow over the vehicle has no viscous correction applied. The vehicle is composed of only a body and a duct.
Chapter 5

DPLL Viscous Drag

Forces on a submerged marine vehicle include both inviscid and viscous contributions. The inviscid forces and the information from those forces is detailed in Sections 3 and 4. The discussion here covers viscous forces.

Viscous forces are composed of friction and residual drag. Friction drag is the tangential force felt on the surface of the body due to viscous shear stresses. Residual drag is composed of form drag and resistance due to the production of surface waves. The vehicles in this work are assumed to be deeply submerged, so the residual drag is considered to be only form drag. Form drag is due to normal pressure forces on the body due to the presence of a boundary layer on the body surface.

Viscous drag on the vehicle bodies in DPLL is assumed to be composed primarily of friction drag [29]. Additional drag due to equipment on the surface of the vehicle is often accounted for with an increment to the friction drag. In the same manner, the small portion of the total drag which is the form drag could be included in the viscous drag calculation.

DPLL uses an input friction coefficient for the calculation of viscous drag on the
vehicle. Future versions of DPLL should be improved with a detailed calculation of the form drag. The friction coefficients described in this section, however, are assumed to be input into the program and to account for all of the viscous drag on the body and duct.

5.1 Centerbody friction drag

5.1.1 Selection of a frictional drag coefficient

As detailed more fully in Section 2.4 the vehicle centerbody in DPLL is represented hydrodynamically by an equal number of control points on the body surface and submerged source rings. The induced velocities of the source rings combined with the ship speed, the induced velocities from the duct singularities, and the induction from the blade-rows produces a flow along the body surface that is tangent to the body surface at each body control point (See Figure 3-2).

This local velocity, $q$, must therefore be the cause of friction drag felt by the vehicle from the water through which it is passing. Globally, the vehicle is moving at ship speed, $V_S$, while the local velocities are those composed of ship speed that has been modified by the effects of all of the components of the vehicle.

A coefficient of friction drag, $C_{FB}$, can be utilized to determine the frictional resistance of the body. This coefficient is one of the inputs to the program and is normalized such that

$$C_{FB} = \frac{Drag_v}{\frac{1}{2} \rho q^2 S}$$

(5.1)

where $Drag_v$ is the dimensional viscous drag on some surface area $S$. In this equation,
$S$ is the area of a given panel at which there is the local control point inviscid velocity, $q$.

The frictional resistance problem could have been organized around a $C_F$ that was normalized by ship speed and the total surface area of the body. But, instead, a $C_F$ based on panel surface area and local velocities was chosen so that the additional frictional resistance felt by the body due to the accelerated flow around the propulsor, if present, could be accounted for by the local velocity dependence of the friction drag on each panel. Additionally, the local velocity contribution will likely make the calculated friction drag on a body different from that same surface area treated with just flat plate friction drag (no thickness). The reason for this is that a body with internal volume causes decelerated and accelerated inviscid flow over different parts of its exterior, unlike a flat plate that should see ship speed everywhere. The associated change in friction drag due to velocity changes over the body shape is accounted for by the influence of the local velocity in the current scheme. The magnitude and direction of these changes to the friction drag of a body are complicated and beyond the scope of the preliminary nature of this work. Therefore, it is expected that user experience or good judgment should dictate the selection of $C_F$ for correct results. Lacking other direction, a user could certainly turn to the ITTC friction coefficient curve which is a function of Reynolds number and can be found in [50].

5.1.2 Calculation of body friction drag

The vehicle is a three dimensional axisymmetric body which changes in radius over its length. The axial boundaries of each panel on a two dimensional representation of the body (see Figure 2-1) allow the three dimensional surface of the body to
be discretized into a number of tubular sections, of finite length and of generally monotonically increasing or decreasing radius over that length. In this work, the boundaries between panels were chosen to be the midpoint between adjacent control point locations. These sections are drawn by intersecting a plane that is perpendicular to the body axis with the body at the axial location of each junction of a panel with its neighbor. Though the exact change of radius in the axial direction is a continuous function, each panel begins to look like it has constant slope (change of radius) along its length as the number of panels along the body increases. Each section can therefore be approximated by the shape of a right circular cone that has had its point truncated by a plane parallel to its base. Shown in Figure 5-1, this geometry's official name is a "frustum of a right circular cone" [51].

![Figure 5-1: Frustum representation of body panels](image-url)
The surface area, $S$, of each frustum section along the body can be calculated [51] from

$$S = \pi \left( R_1 + R_2 \right) s$$  \hspace{1cm} (5.2)

where $s$ is the slant height of the frustum, and $R_1$ and $R_2$ are the radii of each of the two bases. The continuous surface of the body is broken up into discretely sized panel frustums on which the drag coefficient, $C_F$, can be applied to calculate the local drag contribution from each panel. As this drag is the summation of axial forces, the force from each frustum must be adjusted to account for only the axial portion of the drag force. In the case of the example frustum shown in Figure 5-1, this amounts to multiplying the surface area of the frustum by the cosine of the angle $\theta$ where

$$\theta = \arctan \left( \frac{|R_1 - R_2|}{|X_1 - X_2|} \right)$$  \hspace{1cm} (5.3)

The summation is thus written

$$Drag_v = \sum_{i=1}^{n} C_F \frac{1}{2} \rho q_i^2 S_i \cos \theta_i$$  \hspace{1cm} (5.4)

where $Drag_v$ is the total viscous drag on the body, $n$ is the number of frustum panels on the body, $C_F$ is from Equation 5.1, $q_i$ is the local velocity tangent to the panel surface, and $S_i$ is the surface area of the each frustum panel in question.

This dimensional summation of the frictional drag experienced by the body surface is converted into a drag coefficient with a normalization identical to that of the other drag coefficients in this work. This new friction drag coefficient for the friction drag
of the entire vehicle centerbody is written

\[ C_{DV} = \frac{\text{Drag}_V}{\frac{1}{2} \rho V_S^2 A_o} \]  \hspace{1cm} (5.5) \]

where \( A_o \) is the maximum cross-sectional area of the vehicle body and \( V_S \) is the ship speed.

### 5.2 Duct friction drag

As shown in Figure 5-1, an axisymmetric body can be broken up into sections by planes that are perpendicular to the body axis. As with the centerbody, the duct can be broken up into a number of frustums of right circular cones.

The calculation of the friction drag on the duct surface is analogous to the calculation of the friction drag on the body surface in Section 5.1. The duct friction drag calculation is based upon an input friction coefficient that is prescribed separately from the body friction drag coefficient because the duct and body are typically operating at very different Reynolds numbers. The details regarding the construction of frustums and the drag summation over the duct are identical to that for the body except that the duct has an inner and outer surface while the body necessarily has fluid flow only on the outside. The drag on each section of the duct is the summation of that on the outer and inner duct surfaces.

The singularity representation of the duct is with vortex rings along the duct meanline surface. If there are \( n \) vortex rings of varying discrete strength \( \Gamma_i \) on \( n \) panels, then a vortex sheet strength, \( \gamma_i \), can be assigned to each panel where
\[ \gamma_i = \frac{\Gamma_i}{s_2 - s_1}. \] (5.6)

and \( s_2 - s_1 \) is the axial arclength along each panel. From Section 4.2, a sheet strength of \( \gamma \) represents a jump in velocity across the surface of the duct of magnitude \( \gamma \). In this case, we have a constant sheet strength, \( \gamma_i \), on each panel so that the resultant velocity for the upper(+) and lower(-) sides of the duct at panel \( i \) is

\[ u^\pm = u_i \pm \frac{1}{2} \gamma_i. \] (5.7)

In summary, the friction drag on the duct surface is calculated discretely on each frustum panel and summed over both sides of the entire duct. This total dimensional friction drag experienced by the duct surfaces can then be converted into a drag coefficient with a normalization that is identical to that of the other drag coefficients in this work. This new frictional drag coefficient for the viscous drag on the entire duct is written

\[ C_{DV} = \frac{Drag_V}{\frac{1}{2} \rho V_s^2 A_o}. \] (5.8)

where \( A_o \) is the maximum cross-sectional area of the vehicle body and \( V_s \) is the ship speed.

\[ \text{5.3 Appendage drag coefficient} \]

The drag contribution of the appendages to the vehicle, if any, must be accounted for. As the present work is focused upon the propulsor, duct, and body, the appendage drag here is accounted for with a single input drag coefficient, \( C_D \). The drag, \( Drag_A \),
accounts for all drag (pressure, friction, form) contributions from the appendages and is normalized on ship speed and body maximum cross-sectional area.

\[ C_D = \frac{Drag_A}{\frac{1}{2} \rho V_S^2 A_o} \]  \hspace{1cm} (5.9)

The normalization of this drag coefficient is identical to that of the other calculated force coefficients for the vehicle, which allows the direct comparison of all force coefficients.

### 5.4 Overall drag coefficient for vehicle components

The individual total drag coefficients for the body, duct, and appendages each includes the inviscid and the viscous drag for that component. The total body and total duct drag coefficients are constructed by adding together the viscous \( C_{Dv} \) and the inviscid \( C_{Dp} \) drag coefficients of each such that

\[ \text{Body } C_D = \text{Body } C_{Dv} + \text{Body } C_{Dp} \]  \hspace{1cm} (5.10)

\[ \text{Duct } C_D = \text{Duct } C_{Dv} + \text{Duct } C_{Dp} \]  \hspace{1cm} (5.11)

The total appendage drag coefficient (Appendage \( C_D \)) is input in a form that includes both the inviscid and viscous contributions.

A total drag coefficient, \( C_{DT} \), can be constructed for the vehicle which includes the total drag coefficients of the body, duct, and appendages:

\[ C_{DT} = \text{Body } C_D + \text{Duct } C_D + \text{Appendage } C_D. \]  \hspace{1cm} (5.12)
Figure 5-2: Body, duct, and total $C_D$ for unpropelled vehicle vs. ship speed. The flow over the body has no viscous correction to the profile. The duct is specified to have zero net circulation.

None of the above $C_D$ definitions contain any propulsor forces.

5.5 Duct-body friction $C_D$ results vs. ship speed

The total appendage $C_D$ is an input and is therefore constant. The trend of total $C_D$ for the body and total $C_D$ for the duct, along with the resultant total drag coefficient, can be examined versus ship speed. For these results, the vehicle is composed of a body and duct and the input appendage total $C_D = 0.001$. The duct is specified to carry zero circulation and therefore will have very little effect on the pressure contribution of the duct and body (Detailed in Section 4.5). Shown in Figure 5-2 is the total drag coefficient for this no-load duct vehicle, with no propulsor, for various ship speeds with uniform inviscid inflow.
The curves of the non-dimensional $C_D$ vs. ship speed are flat for these results. This is an indication that the normalization and combination of all of the drag coefficients are correct in this work. A more realistic result might have shown $C_D$ for each component, and thus the total, to be a function of speed due to the fact that the friction coefficient really should change with speed and Reynolds number. It does not change in this work, and a reference to its improvement is therefore included in the section of future work and research in this area. The constant linear results shown in Figure 5-2 were typical of those obtained with a number of body shapes varying from a very highly tapered body to a straight shaft of constant diameter.

5.6 Duct-body friction $C_D$ results vs. duct circulation

Section 4.6 explains the expected trends in the drag coefficients of the body and duct as the duct circulation is varied. The reasoning for drag coefficient changes with duct load variation is identical here, but in this case the same analysis is carried out including frictional drag on the body and duct. Figure 5-3 shows the variations in vehicle friction $C_D$ as the duct circulation is varied.

As with the inviscid analysis, the variations for the friction case show increased body drag for an increased duct thrust, and vice versa. In general, though, the overall vehicle $C_D$ remains about constant. In the example shown here, there does appear to be a downward trend in overall vehicle $C_D$ with increasingly negative duct circulations. This trend was deemed negligible, but illuminated a point with duct load limitations. In looking more carefully at the results for a duct circulation of $-0.1$, it is clear that the necessary duct shape for that amount of load is of camber so extreme
Figure 5-3: Body, duct, and total $C_D$ including friction forces for an unpropelled vehicle vs. duct circulation. The flow over the body has no viscous correction to the profile. The vehicle is composed of only a body and a duct.

that there is no way for the designer to expect that such a configuration is physical. Therefore, Figure 5-4 served as warning to also look carefully at the geometry output for the vehicle, rather than just the resulting force trends.
Figure 5-4: Example of duct with extreme negative (clockwise) circulation to illustrate limits to the amount of load that a duct can be expected to carry in an effort to change overall powering of vehicle.
Chapter 6

DPLL Duct Design Procedure

This section covers the design of the duct shape and angle of attack within DPLL. The magnitude of the duct camber and angle of attack are solved iteratively under the constraint of a user specified duct circulation.

As detailed more fully in Section 2.3, DPLL represents the propulsor duct mean camber line by vortex rings. The vortex rings and control points are cosine-spaced in arclength along the meanline. Part of the input to DPLL is the camber shape to be used for the duct mean camber line. This shape is a B-spline curve input as B-spline vertices. The vertices are adjusted during each duct design iteration to satisfy the kinematic boundary condition at each one of the duct control points. This adjustment is made by way of changes to the duct angle of attack and camber.

6.1 A hypothetical duct design

A description of the iterative procedure for the design of the duct meanline shape within DPLL is illustrated here for a hypothetical case of a duct represented by 5
vortex rings along the duct meanline curve. Therefore, there are also 5 control points positioned along the meanline.

Each iteration of the design results in a 3-part solution which satisfies the kinematic boundary condition of zero normal velocity at each one of the duct control points. These three parts to the solution are listed here.

1. Strength of duct vortex rings, $\Gamma_i$

2. Increment to current duct angle of attack, $\alpha$

3. Increment to current duct camber, $f$

6.2 Position of governing blade-row tip

As part of the duct design, one point on the duct needs to be fixed in space. This point is the intersection of one of the propulsor blade-row tips with the duct meanline. This point, termed the "governing blade-row tip", is an input to the problem. The governing blade-row will usually be a rotor as a rotor's diameter and tip axial location are often quite well defined at the problem outset based upon the designer's needs. Any one of the blade-row tips can be selected, though, as the duct design just needs a particular point along the duct chord to fix in space in order to guarantee that the solution is determinate. Otherwise, without a radial constraint, appropriate solutions to the duct design could exist at multiple radii.
\[
\begin{bmatrix}
A_{11} & A_{12} & A_{13} & A_{14} & A_{15} & A_{16} & A_{17} \\
A_{21} & A_{22} & A_{23} & A_{24} & A_{25} & A_{26} & A_{27} \\
A_{31} & A_{32} & A_{33} & A_{34} & A_{35} & A_{36} & A_{37} \\
A_{41} & A_{42} & A_{43} & A_{44} & A_{45} & A_{46} & A_{47} \\
A_{51} & A_{52} & A_{53} & A_{54} & A_{55} & A_{56} & A_{57} \\
1 & 1 & 1 & 1 & 1 & 0 & 0 \\
DX^2 & -DX^1 & 0 & 0 & 0 & 0 & 0
\end{bmatrix}
\]

Figure 6-1: Duct design matrix for 5 vortex ring example

### 6.3 Initial duct conditions: Analysis

The first step in the design of the duct is to perform an analysis of the entire input vehicle to get a first guess at the normal velocities at the control points on the input duct meanline. This input vehicle could be just a duct, or a duct with a body, or a duct with a body and blade-rows. In the case of a duct design without the presence of a propulsor, a governing blade-row tip location must still be specified to govern the location of one point on the duct in order to insure a unique duct shape solution.

### 6.4 Construction of duct design matrices

For the design of a duct composed of \( n \) vortex rings (and thus \( n \) control points), the coefficient matrix is a square matrix of \((n + 2)\) by \((n + 2)\) entries. The coefficient matrix for this example of 5 vortex rings is shown in Figure 6-1.

Each of the top five rows of Figure 6-1 is an equation for the normal velocity induced at each duct control point due to all of the vortex rings (Section 6.4.2). The final two rows are the specifications for total duct loading (Section 6.4.3) and finite leading-edge loading (Section 6.4.4), respectively.

The column matrix solution for the duct design example here would be composed
\[\begin{bmatrix}
\Gamma_1 \\
\Gamma_2 \\
\Gamma_3 \\
\Gamma_4 \\
\Gamma_5 \\
d\alpha \\
df
\end{bmatrix}\]

Figure 6-2: Duct solution column matrix for 5 vortex ring example

of the five discrete vortex rings strengths, $\Gamma_i$, plus a solved increment to the duct angle of attack, $d\alpha$, and a solved increment to the duct camber magnitude, $df$.

### 6.4.1 RHS column matrix

The right-hand side column matrix (RHS) in Figure 6-3 contains the value of the normal velocity at each of the duct control points. This normal velocity includes the free stream velocity due to ship speed, the induction due to the presence of the vehicle center body, and induced velocities due to the presence of the blade-rows. These normal velocity components are updated with each iteration through the program as the strength of the body singularities and the geometry and load distribution of the blade-rows are also iterated.

In this example with five vortex rings, the first five entries in the RHS are the normal velocities, $V \cdot n_i$. The sixth entry is the user-specified value of the total duct circulation

$$
\Gamma_{DUCT} = \sum_{i=1}^{N} \Gamma_i
$$

where $N$ is the number of duct vortex rings and $\Gamma_i$ is the strength of each vortex ring.

The seventh entry in this example is a zero which allows Equation 6.4 to be con-
\[
\begin{bmatrix}
V \cdot n_1 \\
V \cdot n_2 \\
V \cdot n_3 \\
V \cdot n_4 \\
V \cdot n_5 \\
\Gamma_{DUCT} \\
0
\end{bmatrix}
\]

Figure 6-3: Duct RHS column matrix for 5 vortex ring example

structured for the specification of finite load at the duct leading edge.

6.4.2 Duct-on-duct influence functions

The upper left-hand $5 \times 5$ entries in the matrix in Figure 6-1 are influence functions for the normal velocity induced at each one of the control points (rows) by each of the vortex rings (columns), if each vortex ring were of unit strength. The normal to the duct surface at each control point is pointed toward the inside of the duct.

Figure 6-4: Normal vector to the duct meanline
6.4.3 Specification of total duct loading

The sixth row of the matrix in Figure 6-1 is the line which specifies the total duct loading. The first five entries (columns which correspond to the five vortices in this example) are all assigned the value of 1. These unit entries represent an equation for the summation of the discrete strength of each of the duct vortex rings.

6.4.4 Specification of finite leading edge load

The seventh, and last, row of the matrix specifies finite loading at the leading edge of the duct. In order to design a duct which is at or near ideal angle of attack, the duct vortex sheet strength approaching the leading edge of the duct must extrapolate to a small finite number. In this work, the sheet strength at the leading edge is based upon an extrapolation of the sheet strengths on the first two panels.

As shown in Equations 4.1 and 4.2, the sheet strength of a particular panel on the duct is constant. It is equal to the discrete vortex strength for that panel divided over the arclength of the panel. The duct panels are divided into lengths according to the spacing of Lan [44], allowing the conversion of discrete vortex strength into sheet strength. Accordingly, the strengths on the first and second panels of the duct leading edge can be expressed, respectively, as

\[ \gamma_1 = \frac{\Gamma_1}{DX1} \]  

(6.2)

and

\[ \gamma_2 = \frac{\Gamma_2}{DX2} \]  

(6.3)
where

\[ \Gamma_1 \text{ and } \Gamma_2 \text{ are the strengths of discrete vortices 1 and 2} \]

and \( DX_1 \) and \( DX_2 \) are the lengths of panels 1 and 2.

The last row of the design matrix (Figure 6-1) is filled with zero entries except for columns 1 and 2. Column 1, entry \( A_{71} \) in this example, is assigned \( DX_2^2 \), the square of length of the second panel. Column 2, entry \( A_{72} \), is assigned \(-DX_1^2\), the negative square of the length of the first panel. When combined with the solution matrix (Figure 6-2) and the RHS matrix (Figure 6-3), this leads to the equation

\[ \Gamma_1 DX_2^2 - \Gamma_2 DX_1^2 = 0. \]  \hspace{1cm} (6.4)

This can be rewritten using Equations 4.1 and 4.2 as

\[ \frac{\gamma_1}{DX_1} = \frac{\gamma_2}{DX_2} \] \hspace{1cm} (6.5)

where \( \gamma_1 \) and \( \gamma_2 \) are the vortex sheet strengths of the first and second duct panels, respectively. Equation 6.5 forces the sheet strength at the duct leading edge to be finite and small because panel size is decreasing in the direction of the leading edge.

### 6.4.5 Duct normal velocity due to duct angle of attack

Column 6 in the duct design matrix, Figure 6-1, represents a change in the normal velocity at each of the duct control points due to a small perturbation of the duct angle of attack, \( \alpha \), about the governing blade-row tip (see Section 6.2). The first five entries \((A_{16} - A_{56})\) in this column are filled with the change in normal velocity while
the last two entries ($A_{66}$ and $A_{76}$) are each filled with zero. Column 6 adds an extra term to the equation for the normal velocity at each duct control point. This extra term allows for the solution of an increment to the duct angle of attack.

6.4.6 Duct normal velocity due to duct camber

Column 7 in the duct design matrix, Figure 6-1, represents a change in the normal velocity at each of the duct control points due to a small perturbation of the duct camber magnitude, $f$. The first five entries ($A_{17}$-$A_{57}$) in this column are filled with the change in normal velocity while the last two entries ($A_{67}$ and $A_{77}$) are each filled with zero. This column adds an extra term to the equation for the normal velocity at each duct control point. This term allows for the solution of an increment to the duct camber.

6.5 Problem solution and iteration of duct design

6.5.1 Matrix solution

With the duct design matrix (Figure 6-1) and the right-hand-side matrix (Figure 6-3) constructed, the strength of the duct vortex rings and the increments to the duct angle of attack and camber can be solved. The solution matrix is obtained using upper-lower decomposition of the duct design matrix (Figure 6-1).

The duct design process is iterative in nature because the duct design matrix is constructed necessarily from the present duct shape. Therefore, zero normal velocity at each of the duct control points is attained by the solution of the vortex rings strengths for the previous geometry but reflecting help from the solved increments to
angle of attack and camber.

6.5.2 Duct design iterations

Upon solution, the duct must now be repositioned in the problem to reflect the new angle of attack and camber. Additional iterations of this process must take place before the solved increments to angle of attack and camber, and therefore also changes to the vortex rings strengths, tend to zero.

The inclusion of the duct design process into the entire design routine is likewise iterative due to the proximity of all of the parts of the vehicle to each other. That is, a newly designed duct will change the flow “seen” by the vehicle center body and the blade-rows. Thus, upon completion of the duct design, the singularity strengths of the other parts of the vehicle must be updated to reflect the new duct geometry. This updating also holds for the duct and propeller when the body is re-solved and the body and duct when the propellers are re-designed.

6.5.3 Duct radius adjustment

As introduced in Section 6.2, the duct design process must include one point on the duct, at a specified linearized chordwise location on the duct, which is fixed relative to the vehicle center body and the global coordinate system. Upon completion of the duct design process, there is no guarantee that the duct will still intersect that fixed point due to change in the absolute camber of the foil shape. The change in $\alpha$ is forced to occur around the fixed point and so will not affect the intersection, but a change in camber can place the duct surface at that linearized chord location either above or below the desired fixed point. To rectify this problem, a process was
incorporated into the duct design iterative process which increases or decreases the duct radius appropriately to replace the duct so it intersects the fixed point of the governing blade-row tip.
Chapter 7

Lifting Line Design Procedure

Following the duct design process in DPLL, representations of the propulsor blade-rows must be added to the flow field in the region between the designed duct and the vehicle afterbody. As described in Section 2.5, each of the blade-rows in DPLL is represented by a lifting line composed of a finite number of straight vortex segments. Each individual segment has constant circulation strength along its length. Each vortex segment also has a control point which is centered between the vortex ends. The rationale for central spacing of control points on each vortex segment is detailed in Section 8.7. This chapter discusses the iterative design of the lifting line blade-rows in DPLL.

7.1 A hypothetical design of blade rows

7.1.1 Background

An example of the iterative procedure for the design of the lifting line blade-rows in DPLL is described here for a propulsor with two blade-rows. This hypothetical
design is a forward fixed stator providing a swirled inflow to a downstream right-hand-rotating blade-row. The stator swirl is opposite the rotation of the rotor. The description of the procedure for this example can be extended to a propulsor with a greater or lesser number of blade-rows.

Calculations involving a stator differ from those for a rotor only in the sign and magnitude of the input advance coefficient. The sign of the advance coefficient, positive or negative, indicates the direction of rotation of each blade-row. A positive advance coefficient corresponds to a right-hand blade-row and conversely for a negative advance coefficient. For a stator, the input advance coefficient is some large magnitude number which indicates to the program that the blade-row is essentially non-rotating (see Section 2.5.2 for the definition of advance coefficient). The swirl direction of a stator is indicated by the sign of the advance coefficient. Right-hand stator swirl is from a positive advance coefficient. The velocity calculations involving rotation rate are identical in form between rotating and fixed blade-rows except that \textit{DPLL} uses a zero rotation rate for the stator and extracts the rotation rate from the advance coefficient for a rotor.

The result of each iteration of the propulsor design process is the lifting line geometry of the propulsor, the trailing vortex geometry, and the calculated circulation distribution on each blade-row. From these results, the induced velocity field can be calculated and a new iteration of \textit{DPLL} can be started. A flowchart of this procedure is shown in Figure 10-4.

\footnote{A right-hand blade-row rotates clockwise when viewed from downstream. Left-hand rotation is counter-clockwise.}
7.1.2 Identification of a governing blade-row tip

As part of the blade-row tip information input to DPLL, one of the blade-rows of the propulsor must be input as the “governing” blade-row. This choice is simple for a single blade-row propulsor. For a multiple blade-row propulsor it tends to be the designer’s choice. The governing blade-row is the only blade-row of the propulsor for which the designer defines a fixed tip radial and axial position. For the example here, the rotor will be the governing blade-row.

There is an assumption in DPLL that all blade-row tips have “zero-gap” where the tip of each blade-row is assumed to end in contact with the duct, leaving no radial gap. A fixed governing blade-row tip that lies on the duct surface guarantees a determinate solution to the duct design process explained in Section 6.2.

7.1.3 Growing lifting lines

When the blade-row lifting line process is reached in DPLL, the duct has been designed and passes through the fixed governing blade-row tip location. The linearization chordwise position on the duct of the intersection of each of the blade-rows with the duct surface is known from input to DPLL. Therefore, a single point is known of the geometry of each blade-row. These points are the tips’ axial and radial coordinates.

Blade-rows other than the governing blade-row are defined only by the axial locations of the intersections of their tips with the duct. As the duct position and shape changes during DPLL, the radial positions of tip intersections with the duct vary with the duct radius change at the tip axial locations. As the duct design iterations vary the duct angle of attack and camber, the radius of non-governing blade-rows tips
are likely to vary from one lifting line design iteration to the next until convergence of the entire design.

The process by which the geometries of lifting lines are determined in DPLL is called “growing” the lifting lines. In this example, the stator is the most upstream blade-row. From its tip location, a piecewise continuous line is “grown” in from the duct to the body. As this line is grown in small linear increments, the velocity at each new point is evaluated to determine a direction normal to the flow in which the next small step is taken. Eventually, the grown stator lifting line will intersect the body, determining the axial and radial location of the hub for the stator.

A lifting line for the adjacent blade-row downstream, in this case the rotor, is next grown from its tip location on the duct into the body. If a third stage were present, it too would be grown from duct to body.

### 7.2 Spacing lifting lines

The grown piecewise continuous line for the geometry of the first upstream blade-row, in this case the stator, is represented with two sets of cubic spline coefficients. The coefficients represent the axial position of the line versus arclength along the line and radial position of the line versus arclength along the line. This cubic spline representation is then evaluated at constant arclength intervals in order to break the line up into uniformly spaced segments of the curve. These break points then are used as the endpoints of straight vortex segments of nearly uniform length which are used in the lifting line representation of the stator.

Clearly, a highly curved line will be poorly represented with just a few straight
lines, but the gradients of the axial and radial flow velocities due to the total induction of body, duct, and propulsor are quite moderate in DPLL and lead to relatively straight grown lines. Therefore, the straight vortex segments are an adequate representation of the curved grown line. This approximation improves for higher numbers of vortex segments.

7.3 Trailing vortices

From Equation 14.1, vortex segments cannot end in the flow and must therefore shed trailing vortices from each end. Thus, the stator must have trailing vortex lines which trail aft from the ends of each of its spanwise vortex segments. Vortices trailing off of the stator convect in the axial and radial directions with the local flow and intersect the downstream rotor’s grown line.

This intersection of lines breaks the downstream rotor grown line up into the same number of pieces as the stator. This allows straight vortex segments to be positioned between the intersection points on the rotor so that both the stator and rotor are represented with equal numbers of straight spanwise vortex segments. The axial and radial trajectories of the trailing vortices do not break the downstream blade-rows up into segments of identical length, but they turn out to be very close to uniform in length.

Next, the trailing vortex lines between adjacent blade-rows and aft of the last row up to a specified axial location of the ultimate wake are all broken up into the same number of equally spaced trailing vortex segments. Thus, the axial, radial, and tangential location of all of the lifting line segments are known (the blade-rows are
assumed to have no skew in \textit{DPLL}) and the axial and radial positions of the ends of each trailing vortex segment are likewise known.

### 7.4 Distribution of blade-rows induced velocities

For the initial iteration of the program, the presence of the blade-rows has not been accounted for. In that case, the local velocity used for growing lifting lines is calculated only from ship speed and the induction of the singularities representing the duct and the body as determined from the body—duct analysis (Figure 10-3).

At the end of the first program iteration, however, an initial geometry of the blade-rows and initial distributions of circulation have been calculated. From Equation 14.1, the strength of all trailing vortex segments are also known. The velocity calculation described here is the summation of velocities induced by all of the blade-row spanwise vortex segments plus those induced by the trailing vortex segments.

Three-dimensional induced velocities cannot be calculated without knowing the three-dimensional locations of each end of each trailing vortex segment. Only the axial and radial coordinates of the ends of each trailing vortex segment are known.

The tangential components of the coordinates of a given trailing vortex segment are determined by using the circumferential mean tangential velocity at the segment. The tangential velocity determines the circumferential distance traveled by the vortex segment during its convection from the axial location of its upstream end to the axial location of its downstream end. The axial and radial locations are known for the upstream end of the vortex and an arbitrary circumferential location can be selected. The axial and radial locations are known for the downstream end and the change in
circumferential distance at the downstream end can be added to the upstream circumferential location to calculate the downstream circumferential coordinate. Therefore, three-dimensional coordinates can be calculated for the ends of every trailing vortex segment.

With the three-dimensional geometry of the entire lifting line propulsor and trailers known, the calculated circulation distribution on each lifting line can be used to determine the propulsor induced velocity at any point in the flow field. This induced velocity field is stored in the velocity grid (Section 8.9.2). The stored induced velocity field from one iteration of DPLL also allows the next blade-row design iteration to take place in a flow field that also includes the induced velocities of the propulsor. In addition, the propulsor induced velocities on the duct and body are calculated so that the strength of the singularities representing those components can be updated in the next iteration of the program to reflect the induced velocity effect of the propulsor.
Chapter 8

DPLL Calculation of

Propeller-Induced Velocities

When the lifting line blade-rows and trailing vortex geometry have been added to the problem, calculation of the Goldstein factors for this geometry leads to the calculation of the circulation distribution on each blade row. Based on Kelvin’s theorem (Equation 14.1) the strength of all of the trailing vortices are also known. This section discusses the calculation of the velocities induced by the blade-rows’ vortex systems.

8.1 Lifting line representation of a propeller blade

Velocities induced by a lifting line are due to the lifting line’s bound and trailing vortices. The continuous distribution of circulation along the span of the lifting line is broken up into discrete bound vortices of constant strength along each segment. Since the spanwise loading is now made up of discrete jumps, the free vortex sheet of continuous strength across the span can likewise be replaced by a discrete representation
made up of concentrated vortex lines shed from the end of each bound vortex. Stated another way, the continuous distribution of circulation across the span is replaced with a set of discrete horseshoe vortices, each consisting of a bound vortex segment plus two concentrated trailing tip vortices having the same circulation magnitude as the bound vortex.

8.2 Self-induced velocities

For the calculation of induced velocities from such a discrete horseshoe problem it is necessary to calculate “horseshoe influence functions” for the induced velocities at each field point due to each horseshoe. The influence functions, calculated assuming unit bound vortex strength, are then multiplied by the actual strength of the bound vortex for that horseshoe to obtain the field point velocity.

Real propellers are finite-bladed and will therefore produce a velocity field that varies circumferentially in a coordinate system that rotates with the propeller. Likewise, the global coordinate system sees a velocity field that varies both spatially and temporally due to blade-row rotation.

8.3 Circumferential mean induced velocities

Once the trailing vortices of a given horseshoe have convected downstream beyond about two propeller diameters, the induced velocities at the propeller plane due to those vortices is mostly circumferentially constant. This circumferentially constant effect tends to occur sooner (from vorticity less far downstream) with increasing number of blades and/or decreasing hydrodynamic angle at which the wake leaves
the blades and convects downstream.

8.4 Velocities induced by other blade-rows

$DPLL$ and $PBD-14$ are steady flow models based on time-averaged velocities. Assuming a non-zero relative rotation rate between two blade-rows, the time averaged velocity induced on one blade-row by the other is the circumferential mean velocity in the coordinate system of the affected blade-row.

Due to the tendency of the flow field for a hubbed and ducted propulsor to be largely circumferentially constant, a time averaged approach is an appropriate approximation for the present work. This approximation is sufficient for the row-to-row “interaction” velocities for steady problems such as those addressed in $DPLL$ [9].

8.5 Distribution of control points on the lifting line

Forces on the lifting line are calculated using the Kutta-Joukowski theorem (see Equation 9.14). Assuming the circulation of an individual vortex segment is known, the correct force result necessitates an accurate determination of the velocity at that segment. As detailed in [37], the location of the control point for velocity calculation on each vortex segment is important for accurate self-induced velocity calculations. It is well established that the use of cosine spacing for both the vortex segment endpoints and the control point locations on the lifting line gives accurate velocities and resultant accurate forces [43].
8.6 Control point location for velocities induced by other blade-rows

In the case of a multiple blade-row propulsor, the trailing vorticity of upstream blade-rows will be convected through and past downstream rows. In Section 8.5, it was established that for accurate self-induced (non-circumferentially constant) velocity calculations, those trailing vortices should be leaving a given blade-row at cosine spaced intervals. As those vortex segments (arranged as helical vortices) are convected downstream, there is no guarantee that they will remain cosine spaced relative to each other nor relative to the boundaries of the centerbody and duct. Therefore, a spacing scheme is needed for the vortex segments and control points on a given blade-row when considering the effect of rows both upstream and downstream of the row in question.

Trailing vortices will intersect downstream blade-rows at certain radial locations, depending upon how the vorticity was convected, and break those blade-rows up into the same number of vortex segments as the upstream blade-row. It is possible that this segment spacing on the downstream row will end up close to cosine spaced, but even small variations from cosine spacing will cause induced velocities, if one were to use this new spacing for the downstream blade-row self-induced calculations, that will differ to some degree from self-induced velocities calculated using cosine spaced vortex segments. Thus, spacing based upon the intersection of convected trailers cannot be used for self-induced non-circumferentially constant velocity calculations. However, as detailed in Section 8.7, this spacing is in fact satisfactory for the calculation of circumferential mean velocities on downstream blade-rows. The calculation
of circumferential mean velocities is relatively insensitive to the details of spacing for both self-induced velocities and velocities induced by other blade-rows.

8.7 Control point and lattice spacing for CMV calculations

To determine an appropriate location for a control point on each spanwise vortex segment, circumferential mean velocities due to a constant pitch helical horseshoe were calculated at various field point locations(Figure 8-1). The location of the field point was varied along the span of the bound vortex to determine the sensitivity of velocity calculations due to field point location. Figure 8-2 shows that the position of the field point can be varied along the bound vortex segment anywhere within the center half of the segment span without any significant change in the axial induced velocity calculated at the point.

A horseshoe vortex induces a velocity which is normal to a helical surface whose pitch matches its own. Therefore, because the axial induced circumferential mean velocity is constant in the vicinity of the middle of the vortex segment (Figure 8-2) , the tangential velocity calculation must also be well behaved and vary only as the inverse of radius. This insensitivity to field point position allows the position of control points for circumferential mean velocity calculations to be at the midspan of each vortex segment.
Figure 8-1: Field point locations for circumferential mean velocity calculations due to a constant pitch helical horseshoe vortex.

Figure 8-2: Plot of DPLL circumferential mean axial induced velocities due to a horseshoe vortex with trailing vortices at r/R=0.4 and r/R=0.8. The control point radius was varied along this segment from r/R=0.41 to r/R=0.79.
8.8 Circumferential mean velocity induced by

a lifting line propeller

A technique for the calculation of circumferential mean velocities induced by a three-
dimensional vortex segment was developed by Buchoux [6]. This technique was utilized in this work because it showed a factor of seven decrease in calculation time when compared to a previous technique developed by Kerwin [32]. This previous technique was contained in the subroutine CMVSEG. The CMVSEG technique involved a Romberg iteration of Biot-Savart's law where the circumferential mean velocity induced by a single vortex segment was approximated by the velocity induced by a large number of segments (of net strength the same as the single vortex segment) obtained by rotating the initial element around the axis of symmetry.

The Buchoux technique is efficient but does not determine influence functions for velocities in the tangential direction. Fortunately, Kelvin's theorem allows us to do so. Consider a spanwise, or bound, vortex segment which is part of an axisymmetric lifting line propeller. Each segment making up a lifting line is assumed to have constant circulation, \( \Gamma \), along its length so that

\[
\Gamma = \frac{2 \pi r \times 2 V_T}{Z} \tag{8.1}
\]

where

\( V_T = \) circumferential mean tangential induced velocity at the vortex segment

and

\( Z = \) number of lifting line blades on the propeller.
A vortex line cannot end in a fluid and so from each end of the vortex segment must be shed a trailing vortex line which is convected downstream along streamlines passing through the ends of the bound vortex segment. Clearly, the constant strength vortex sheet which is shed from this constant strength vortex element is always bounded downstream by the vortices trailing from the ends of the vortex element. By conservation of angular momentum from Kelvin’s Theorem, the time rate of change of circulation within the area bounded by the trailing vortices must be zero. Therefore, the circulation in that region is known, and, from it, the tangential velocity.

Consider a radial vortex segment with a particular constant circulation $\Gamma_i$, along its length and trailing vortices from the segment’s ends that trail downstream along constant radius streamtubes. From Equation 8.1, the circumferential mean tangential induced velocity at the vortex segment, due to that particular vortex element and its trailers, is

$$V_T = \frac{\Gamma_i Z}{4 \pi r}. \quad (8.2)$$

The tangential velocity induced immediately downstream of the vortex segment, and therefore for locations between those two constant-radius trailers anywhere downstream is constant in the axial direction and equal to twice the value at the lifting-line:

$$V_T = \frac{\Gamma_i Z}{2 \pi r}. \quad (8.3)$$

If, on the other hand, the flow downstream of the propeller contracts or expands, the trailing vortices are now no longer of constant radius. The circumferential mean tangential velocity in the region between the two trailing lines is no longer constant
in the axial direction but varies inversely with radius of the streamtube between the segment’s trailers. For a given vortex segment with circulation $\Gamma_i$, the circumferential mean tangential velocity immediately downstream of the lifting-line is given by Equation 8.3 but varies in the axial direction according to the equation

$$V_T = \frac{\Gamma_i Z}{2\pi r} \cdot \frac{r_{new}}{r_{segment}}$$  \hspace{1cm} (8.4)

where $r_{new}$ is the contracted (or expanded) radius of the segment’s trailer streamtube and $r_{segment}$ is the original radius of the vortex segment between those two trailing vortex lines.

8.9 Application of propulsor-induced velocities to the next DPLL iteration

8.9.1 Convergence of the induced velocity field

During a particular iteration of DPLL, the geometry of the trailing vortices and the lifting lines are known (Section 7). Together with the circulation distribution obtained as in Section 2.6.4, the propulsor induced velocities can be calculated at any location. In the next iteration of the program, however, the duct may have moved slightly and the lifting lines may have been “grown” (Section 7) to a slightly different geometry. Clearly, the induced velocity field will be different in this new iteration due to the geometry changes. Moreover, the geometry affects the Goldstein calculation of the circulation distribution, which will further differentiate the present induced velocity field from that of the previous iteration.
Reconciliation of the induced velocity field from one iteration to the next is based upon the eventual convergence of the propulsor and duct geometry with increasing number of program iterations. Upon convergence of the lifting lines' and trailing vortices' geometries, the circulation distribution on each blade-row will also become nearly fixed. Thus it is reasonable to pass the induced velocity field from one iteration to the next with the expectation that at some point the geometry will converge and the passed induced velocity field will be the same from one iteration to the next.

8.9.2 Induced velocity storage grid

The velocity grid is a grid of points (Figure 8-3) located in the center of the boxes created by the trailing vortex segments and the lifting lines.

![Diagram of DPLL velocity grid points for storage of propulsor induced velocities](image)

Figure 8-3: DPLL velocity grid points for storage of propulsor induced velocities

During a particular iteration, the propulsor induced velocities at all of the grid points are evaluated. Thus, a discrete representation of the induced velocity field is created which can be passed on to the next iteration so that the duct and propulsor
design processes can occur in the “presence” of the current propulsor’s induction. Note that the grid points upstream (to the left) of the lifting line are included in the velocity grid so that interpolations from the grid are improved for the evaluation of propulsor induced velocities on the duct. Velocity evaluations at locations falling between the points of the grid can be carried out safely with either an interpolation scheme or some sort of spline of the grid velocities versus grid location. In DPLL, both a linear and a cubic spline interpolation are utilized, depending on the application in the program.

An unfortunate possibility is that the next iteration in the program may call for a velocity evaluation that falls outside of the range of the current velocity grid geometry. As detailed in Appendix A, attempts to interpolate or spline the grid results to locations outside of the grid range can be dangerous. For this reason, in the duct design process (Section 6) and the growing of the new lifting lines and trailers (Section 7), the propeller-induced velocities at a given point for the current iteration are calculated with a linear interpolation from the closet grid points’ locations from the previous iteration.

Though a linear interpolation may be a poor representation of velocities at some distance from the grid, it allows a rough guess at the velocity for the purposes of placing the propulsor and duct geometries. As explained earlier, as the propulsor and grid and duct geometries converge and basically stop changing with increased iterations, the linear interpolation scheme improves as all velocity calculations begin to occur within the grid geometric range, or in very close proximity, to the grid.

Once the new lifting lines and trailers are constructed utilizing the linear inter-
polation scheme for the induced velocities, the grid lattice can be constructed again. At this point, because we don’t yet know the new circulation distributions, the only velocities available to assign to the new grid are velocities from the previous iteration. The induced velocity values from the previous grid are therefore assigned to the newly created grid locations. This way, the new grid is full of velocities for the routine that calculates the influence functions and leads to the new circulation distribution calculation. Once the new circulation distribution is known, the new grid can be filled with the new induced velocities and the next iteration can begin.
Chapter 9

Calculation of Forces for a Lifting Line Propulsor

9.1 Background

The following sections cover the calculation of thrust, torque, efficiency, and the non-dimensional $C_T$, $C_Q$, $C_P$, $K_T$, and $K_Q$ for a lifting line propulsor of diameter $D$ with $Z$ blades. Viscous drag forces are omitted. Traditional definitions of these quantities [17] will be introduced, along with how each relates to the others. Finally, new definitions are given which are tailored to the design of propellers for vehicles with tapered afterbodies.

Most definitions of propeller non-dimensional quantities originated for the description of single, open (no duct) propellers operating in open water conditions where the speed of advance of the propeller, $V_A$, was known and the inflow to the propeller was uniform over the propeller disk. By and large, most of these propeller geometries were quite simple with very small or zero chordlength at the tips of blades and had
very little or no rake and skew. In that case, it was very practical and logical that propeller diameter became a choice for the non-dimensionalization of units of length.

When a duct surrounding the propeller or propellers is employed, very often the chordlengths of the propeller tips are designed to have some finite length. In the case of a cylindrical duct of constant radius and a small gap between the propeller tip and the duct, the diameter of the propeller remains unambiguous. Difficulty regarding the traditional definitions’ dependence on propeller diameter is encountered when the radius of the propeller varies from the blade leading edge to the trailing edge. This is the case for a duct with significant contraction or expansion axially in the proximity of a finite length tip propeller chord. The position at which the propeller diameter can be measured is not unique (tip leading edge, tip mid-chord, or tip trailing edge). Confusion surrounding propeller diameter is compounded for propulsor and vehicle geometries where the body (propeller hub) diameter also varies along the chord of the blade.

The formulation of the design procedure in DPLL makes propeller diameter a poor choice for normalization because the radius of only the governing blade-row tip is necessarily constant. The location of the governing blade-row tip is an input to the process, while the other blade-rows are free to vary in shape and radius during the design. Clearly, non-dimensional coefficients calculated based on variable quantities are useless. The details of the changing radius of a propulsor blade-row can be found in Sections 7.1.2 and 7.1.3.
9.2 New force coefficient definitions

Ducted propulsors on highly tapered afterbodies are the focus of the present work. As described in Section 9.1, quantities which are normalized on propeller diameter are ambiguous for such problems and therefore their re-definition for the problem at hand is warranted.

Force non-dimensionalization of axisymmetric vehicle bodies is often based on the cross-sectional area of the body, $A_B$, calculated from the maximum diameter of the body, $D_B$:

$$ A_B = \frac{\pi D_B^2}{4}. $$

(9.1)

For example, the definition of the body pressure drag introduced in Section 3.2 is

$$ C_{DP} = \frac{Drag_P}{\frac{1}{2} \rho V_S^2 A_B} $$

(9.2)

Based on the $A_B$ precedent, together with the inherent difficulty in assigning a diameter to tapered propulsors, a non-dimensionalization for propeller parameters in this work is developed as follows based on the maximum diameter and cross-sectional area of the axisymmetric vehicle body, $A_B$. In the following sections, the more traditional definitions based on propeller diameter will be termed the "traditional" while the developed ones tailored to highly tapered vehicles and their propulsors will be called the "new".
9.2.1 Advance coefficient

The rate of advance of a propeller relative to its rotation rate is described with the traditional non-dimensional *advance coefficient*, \( J \).

\[
J = \frac{V}{n D},
\]

(9.3)

where \( V \) refers to the axial velocity of the propeller through the water, \( n \) is the rotations per second, and \( D \) is the diameter of the propeller. \( V \) can be either \( V_A \), the velocity of advance, or \( V_S \), the ship’s speed through the water. The corresponding advance coefficients are, respectively, \( J_A \) and \( J_S \). The adopted new definition for advance coefficient is \( J_B \) which is based on ship speed (to avoid the difficult task of defining a velocity of advance for a tapered propulsor) and body maximum diameter (to utilize an unambiguous length).

\[
J_B = \frac{V_S}{n D_B}
\]

(9.4)

9.2.2 Thrust coefficient

The axial force (thrust), \( T \), produced by a propeller is non-dimensionalized in two traditional ways, \( K_T \) and \( C_T \):

\[
K_T = \frac{T}{\rho n^2 D^4}
\]

(9.5)

and

\[
C_T = \frac{T}{\frac{1}{2} \rho V_S^2 A_o}
\]

(9.6)
where \( n \) is the propeller rotations per second, \( D \) is the propeller diameter, \( A_o \) is the disk area of the propeller, and \( \rho \) is the density of water.

Both of these equations depend on propeller diameter and are thus inappropriate for our notional tapered configuration. As with advance coefficient, both definitions could be rewritten with \( D_B \) replacing propeller diameter. As the present work includes options for stators which have a zero rotational rate, even a new \( K_T \) definition based on \( D_B \) would have difficulty with \( n \) equal to zero in the denominator. Therefore, the selected new thrust coefficient is defined based on body maximum cross-sectional area, \( A_B \).

\[
C_T = \frac{T}{\frac{1}{2} \rho \frac{V_s^2}{A_B}} \tag{9.7}
\]

The use of \( K_T \) as a thrust coefficient is discontinued in this work.

### 9.2.3 Torque coefficient

Traditional definitions for a coefficient of torque, \( Q \), necessary to rotate a propeller are

\[
K_Q = \frac{Q}{\rho n^2 D^5}, \tag{9.8}
\]

and

\[
C_Q = \frac{Q}{\frac{1}{2} \rho \frac{V_s^2}{A_o D}} \tag{9.9}
\]

where \( A_o \) is the propeller disk area and \( D \) is the propeller diameter.

For reasoning identical to that in Section 9.2.2 regarding the thrust coefficient
selection, the new torque coefficient is written as

$$C_Q = \frac{Q}{\frac{1}{2} \rho V_S^3 A_B D_B} \quad (9.10)$$

The use of $K_Q$ as a torque coefficient is discontinued in this work.

### 9.2.4 Power coefficient

A power coefficient is often assigned to describe a propeller. The power coefficient, $C_P$, includes $\omega$, the propeller angular rotation rate $2 \pi n$, where $n$ is the propeller rotations per second.

$$C_P = \frac{Q \omega}{\frac{1}{2} \rho V_S^3 A_o} \quad (9.11)$$

The area $A_o$ is the cross sectional area of the propeller disk. Again, the new definition is chosen to avoid dependence on a propeller based diameter and is instead written

$$C_P = \frac{Q \omega}{\frac{1}{2} \rho V_S^3 A_B} \quad (9.12)$$

where $A_B$ is again the maximum cross-sectional area of the axisymmetric vehicle body.

Equation 9.12 can be rewritten by utilizing Equations 9.4 and 9.10 such that it is dependent only on new non-dimensional quantities introduced here:

$$C_P = \frac{C_Q 2 \pi}{J_B} \quad (9.13)$$
9.3 Lifting line force calculations with the new force coefficients

The lift force on an individual vortex element can be calculated by the Kutta-Joukowski theorem

\[ \text{Lift} = \rho U \Gamma \]  \hspace{1cm} (9.14)

where \( \rho \) is the density of the fluid traveling at velocity \( U \) perpendicular past the vortex. The force vector points perpendicular to the inflow velocity \( U \).

For a lifting line propulsor with no rake or skew, the force on a particular vortex element that is part of the propeller will be that due to that element’s circulation and the local velocity “seen” by the vortex segment. As thrust is taken as positive in the upstream direction, the local velocity of concern is the total tangential velocity at the vortex element. Likewise, the torque calculations are based on vortex forces due to the total axial velocity at the element in question.

A traditional definition for the non-dimensionalization of propeller circulation, \( \Gamma \), is

\[ G = \frac{\Gamma}{2 \pi R V_S}, \]  \hspace{1cm} (9.15)

where \( R \) is the propeller radius. The new definition utilized here will instead be

\[ G = \frac{\Gamma}{2 \pi R_B V_S}, \]  \hspace{1cm} (9.16)

where \( R_B \) is the maximum radius of the vehicle.
9.3.1 Thrust

The tangential velocity seen by a vortex element is that due to the propeller's rotation plus the tangential velocities induced by the vortex system of the propeller and its wake. Thus, an incremental measure of thrust, $T$, contributed by the lifting line propeller at radius $r$ is

$$
\delta T = Z \rho \left( \omega r + u_i^* \right) \Gamma \delta r
$$

(9.17)

in which $(\omega r)$ is the rotationally produced velocity, and $(u_i^*)$ is the local tangential induced velocity.

Dividing both sides of Equation 9.17 by $(\frac{1}{2} \rho V_S^2 A_B)$ and using the non-dimensional circulation from Equation 9.16 creates an increment, $\delta C_T$, of our new non-dimensional thrust coefficient, $C_T$,

$$
C_T = \frac{T}{\frac{1}{2} \rho V_S^2 A_B}
$$

(9.18)

where $\delta C_T$ is

$$
\delta C_T = 4 Z \left( \frac{\omega r}{V_S} + \frac{u_i^*}{V_S} \right) G \frac{\delta r}{R_B}
$$

(9.19)

For local velocities and circulation that are functions of radius, the total thrust coefficient of the new form can be expressed as

$$
C_T = 4 Z \int_{R_H}^{R} \left( \frac{\omega r}{V_S} + \frac{u_i^*(r)}{V_S} \right) G(r) \frac{dr}{R_B}
$$

(9.20)

where $R_H$ is the hub radius and $R$ is the propeller tip radius.
9.3.2 Torque

The axial velocity seen by a lifting line propulsor vortex element is composed of the uniform flow due to ship speed plus the axial velocities induced by the vortex system of the propeller and its wake. Thus, an incremental measure of the torque, $Q$, caused by a piece of the lifting line propulsor at radius $r$ is

$$\delta Q = Z \rho (V_S + u_a^*) \Gamma r \delta r$$  \hspace{1cm} (9.21)

where $u_a^*$ is the local axial induced velocity.

Dividing both sides of Equation 9.21 by $(\frac{1}{2} \rho V_S^2 A_B D_B)$ and using the non-dimensional circulation from Equation 9.16 creates an increment, $\delta C_Q$, of our new non-dimensional torque coefficient, $C_Q$,

$$C_Q = \frac{Q}{\frac{1}{2} \rho V_S^2 A_B D_B}$$  \hspace{1cm} (9.22)

where

$$\delta C_Q = 2 Z \left(1 + \frac{u_a^*}{V_S}\right) G \frac{r}{R_B} \frac{\delta r}{R_B}.$$  \hspace{1cm} (9.23)

As with the thrust calculations, local velocities and circulation that are functions of radius can be incorporated into the total new torque coefficient, expressed as

$$C_Q = 2 Z \int_{R_H}^{R} \left(1 + \frac{u_a^*(r)}{V_S}\right) G(r) \frac{r}{R_B} \frac{dr}{R_B}$$  \hspace{1cm} (9.24)

where $R_H$ is the hub radius and $R$ is the propeller tip radius.
9.4 Viscous forces on a blade-row

Viscous forces on each blade-row are considered to be dominated by friction forces. Viscous drag on the blades is calculated from a user-input frictional drag coefficient normalized on blade surface area. A lifting line propeller has no blade area, so an estimate of blade chordlength is also input. Blade-row frictional forces are calculated from the chordlength and the frictional resistance coefficient and resolved into axial and rotational components. These components are added, respectively, to the thrust and torque of the blade-row and therefore included in the thrust and torque coefficients.

9.5 Propeller efficiency

9.5.1 Traditional propeller efficiency

The efficiency of a propulsor has a traditional definition based on $K_T$ and $K_Q$ and $J$. Details of these definitions can be found in [17]. As the $K_T$, $K_Q$, and $J$ definitions depend on propeller diameter, this traditional efficiency definition is ill-defined and cannot be used in the current work.

An additional difficulty with assigning an efficiency to ducted propulsors is that the duct can be made to carry a large axial load as a result of its interaction with the blade-rows. Often the magnitude of this force can be comparable to the thrust on the blade-rows. Clearly, a thrust of this magnitude on the duct would not be present without the blade-rows. This thrust is part of the entire propulsor thrust, but it is difficult to decide where to include the duct thrust and drag for an efficiency
representation of the entire propulsor.

A new definition for propeller efficiency will not be attempted in this work. The reason for this is best illustrated with a comparison between a traditional hull with very small taper and a hull with a highly tapered stern. The gently tapered stern is in general assumed to be streamlined and have negligible or no separation of the flow over the body in the absence of the propulsor. The highly tapered stern, on the other hand, is expected to separate over the afterbody and depends upon the pressure field of the operating propulsor to maintain attached flow in this region.

In the case of the streamlined body there is some thrust deduction, which is typically some small positive fraction of the propeller thrust, such that the towed resistance (unpropelled) of the vehicle is less than that of the propelled vehicle body by the small fraction $t$ times the propulsor thrust necessary for self-propulsion of the vehicle. In effect, the drag of the propelled vehicle has been increased by the low pressure field in front of the propeller acting on the afterbody. This relationship creates a loosely-coupled interaction between the vehicle and the propeller.

In the case of a highly tapered vehicle, where the action of the propeller eliminates the separated flow found when towing the unpropelled vehicle, the thrust deduction factor can be large and negative. Elimination of the separation causes the propelled vehicle drag to drop drastically compared with the unpropelled vehicle drag, hence the large negative thrust deduction. It can be assumed that the power delivered to the shaft will not be significantly different for propelling a streamlined vehicle as compared to propelling a highly tapered vehicle if both vehicles are assumed to have attached flow and similar wetted surface area. Therefore, in the case of the
highly tapered stern, it is easy to envision propulsive coefficients that are well above 1.0 and vary not with actual propeller performance but rather with the degree of body unpropelled separation and the interaction between the separated flow and the propulsor's pressure field. For this reason, the identification of an efficiency for propulsors on highly tapered vehicles will be avoided here.

An additional way to illustrate the problem is to envision a streamlined body with a propeller shaft that extends aft some large distance to the propeller. For this situation, the action of the propeller can be said to be largely free from thrust deduction effects on the hull. If the propeller were adjusted slightly, the effect of that adjustment would be decoupled from any hull effects due to its large distance from the hull. The propeller adjustment would be reflected in the change in efficiency in terms of the traditional definition involving unpropelled vehicle drag, vehicle speed, and power input to the shaft. If instead we could fit a propeller very close to the stern of a highly tapered vehicle, as is the case with vehicles of interest to this work, the action of this propeller will likely decrease the separation of the flow over the afterbody. If this propeller were now to be adjusted just slightly, the change could significantly affect the degree to which the separation problems of the unpropelled hull are rectified by the propulsor. Again, we have illustrated that the comparison of efficiencies between propulsor designs for traditional gently tapered hulls and highly tapered vehicles can be arbitrary and misleading.

9.5.2 Comparison of propulsors based on power consumption

The direct comparison of propulsors for a given vehicle design can be based upon the power coefficient from Section 9.2.4. The power coefficient, $C_P$, can be used as
a direct representation of the power necessary to propel the vehicle with the given propulsor at a given speed.

**DPPL** contains the logic necessary to compute the propelled drag of the vehicle based upon the frictional resistance and Lagally's theorem. The drag (or thrust) of the duct while in operation can also be calculated using frictional resistance and Kutta-Joukowski's theorem. Likewise, the thrust produced by the propulsor can be calculated as described in Section 9.2.2. These three forces are all normalized in an identical way within the program. These three normalized calculations of thrust and drag on the parts of the vehicle while in operation allow the summation of axial forces. A self-propulsion point of the vehicle is that at which the axial forces sum to zero and the body undergoes translation at constant velocity. At this self-propulsion point, the power coefficient can be calculated to represent the power consumed by the propulsor to propel the vehicle at the given speed.

Knowledge of the power necessary for vehicle self-propulsion at a particular speed with different propulsors allows the designer to compare the powering characteristics of different propulsors and propulsor configurations. Likewise, the shape of the vehicle afterbody can be varied by the designer to examine its effect on the powering of the vehicle. The effect of changes in the magnitude of the load carried by the duct on propulsor powering can also be examined to determine optimum distributions of thrust loading between the duct and the propulsor.
Chapter 10

DPLL Program Overview

The flow of logic within DPLL is an iterative pathway through individual modules of the program. The individual modules are the analysis scheme, the duct design scheme, the lifting line design scheme, and the force calculations. The specifics of each module have already been described in detail in previous sections. Therefore, this section will be composed primarily of flow-charts showing the relation and ordering of each of the modules. In order to assist the reader in finding more detail about a particular process, a listing of each process and the section in which it is described is provided in Table 10.1. The overall flow of DPLL is shown in Figure 10-1.
<table>
<thead>
<tr>
<th>Process in \textit{DPLL}</th>
<th>Section in this document</th>
</tr>
</thead>
<tbody>
<tr>
<td>Goldstein reduction factor</td>
<td>Section 2.6.4</td>
</tr>
<tr>
<td>Specification of ( H ) function</td>
<td>Section 2.6.3</td>
</tr>
<tr>
<td>Duct design process</td>
<td>Section 6</td>
</tr>
<tr>
<td>Centerbody inviscid forces</td>
<td>Section 3</td>
</tr>
<tr>
<td>Duct inviscid forces</td>
<td>Section 4</td>
</tr>
<tr>
<td>Centerbody viscous forces</td>
<td>Section 5.1</td>
</tr>
<tr>
<td>Duct viscous forces</td>
<td>Section 5.2</td>
</tr>
<tr>
<td>Lifting-line design process</td>
<td>Section 7</td>
</tr>
<tr>
<td>Lifting-line propeller forces</td>
<td>Section 9</td>
</tr>
</tbody>
</table>

Table 10.1: \textit{DPLL} process description locations

![Flowchart](image-url)

Figure 10-1: Flowchart of program flow within \textit{DPLL}
Figure 10-2: Flowchart for the duct design process
Figure 10-3: Flowchart for the analysis of the body and duct
Figure 10-4: Flowchart for the propulsor design process
Figure 10-5: Flowchart for the force calculations
Chapter 11

Design Applications of \textit{DPLL}

Currently, \textit{DPLL} takes the afterbody shape of the vehicle to be a fixed input to the design process. Though future versions of this work can be envisioned which have some body design process, propulsor design with the current version takes place with the designer varying inputs to the propulsor configuration and parameters only. The primary inputs to the design process are the specifics of the duct shape, size, and load and the specifics of the propulsor. The propulsor specifics include the number of blade-rows, each blade-row's advance coefficient, each blade-row's radial distribution of tangential velocity, and the chordwise location of each blade-row's tip on the duct.

Clearly, there are many parameters to vary in the pursuit of a particular design. It is expected that at least a few items from this list will have some designer prescribed constraints which makes the range of possible combinations smaller. Even so, there remains a large number of example cases which could be pursued to detail the usefulness of \textit{DPLL} as a design tool. A small number of design cases will be undertaken here in which typically one parameter is varied with all others fixed in order to show the resultant powering characteristics of propulsors designed over the tested range of
that parameter.

11.1 Convergence of force results for discretization variations

The first set of tests performed were convergence studies to examine the dependence of vehicle force results upon variations in number of lifting line segments on each lifting line, control points on the body, control points on the duct, and trailing vortex segments in the wake.

Shown in Figure 11-1 is the variation in force coefficients for the vehicle components and the resultant total vehicle force coefficient over a range of number of control points (same as the number of source rings) used to model the body. Above about 125 control points, all force coefficients appear to be unaffected by control point number variations.

Figure 11-2 shows the variation in force coefficients for the vehicle components and the resultant total vehicle force coefficient over a range of number of duct control points (same as the number of vortex rings) used to model the duct.

Figure 11-3 shows the variation in force coefficients for the vehicle components and the resultant total vehicle force coefficient over a range of number of vortex segments used to model the propulsor lifting line blade-row. The divergence of the force results above a certain number of spanwise vortex segments is the result of increasing only the vortex segment parameter and nothing else. It is expected that if other parameters, such as the number of trailing vortex segments, were increased concurrently then larger numbers of spanwise vortex segments could be used with continued convergent force results.
Figure 11-1: Force coefficients for a single blade-row vehicle in uniform inflow with varying numbers of body control points/source rings. The duct is specified to carry no circulation. The inflow to the propulsor has no viscous correction to its profile.
Figure 11-2: Force coefficients for a single blade-row vehicle in uniform inflow with varying numbers of duct control points/vortex rings. The duct is specified to carry a circulation of 0.01. The inflow to the propulsor has no viscous correction to its profile.
Figure 11-3: Force coefficients for a single blade-row vehicle in uniform inflow with varying numbers of vortex segments on the lifting line. The duct is specified to carry a circulation of 0.01. The inflow to the propulsor has no viscous correction to its profile.

Figure 11-4 shows the variation in force coefficients for the vehicle components and the resultant total vehicle force coefficient over a range of number of vortex segments used to model the propulsor trailing vortices. The divergence of the force results above a certain number of trailing vortex segments is the result of increasing only the trailing vortex segment parameter and nothing else. It is expected that if other parameters, such as the number of spanwise vortex segments, were increased concurrently then larger numbers of trailing vortex segments could be used with continued convergent
force results.

Figure 11-4: Force coefficients for a single blade-row vehicle in uniform inflow with varying numbers of vortex segments used to model the trailing vortices. The duct is specified to carry a circulation of 0.01. The inflow to the propulsor has no viscous correction to its profile.

11.2 Single blade-row propulsor: Vehicle force balance vs. advance coefficient

Figure 11-5 shows the variation in force coefficients for the body, duct, and propeller for a single blade-row propulsor at a range of advance coefficients. The duct is given a moderate, but constant, load for all advance coefficients. The series of advance
coefficient variations were carried out for uniform inflow to the vehicle with no viscous corrections to the inflow profile to the propulsor. Also included in the graph is the total drag coefficient for the vehicle, which is the sum of the drag coefficients on the body and duct plus the thrust provided by the propeller. Note that the drag coefficient for the duct is negative, which means that it actually is providing thrust for the vehicle. The advance coefficient where the total drag coefficient is zero is the point at which the vehicle is considered to be self-propelled because all of the axial forces on the vehicle cancel and the vehicle is assumed to undergo constant velocity translation. Total drag coefficients which are less than zero correspond to operating points where drag on the vehicle is greater than thrust. The opposite is true for total drag coefficients which are greater than zero. The self-propulsion point is indicated in the figure.

11.3 Single blade-row propulsor: Power coefficient vs. duct loading

Figure 11-6 shows the variation in power coefficient, \( C_p \), necessary for vehicle self-propulsion over a range of duct circulations. The series of duct circulation variations were carried out for uniform inflow to the vehicle with no viscous corrections to the inflow profile to the single blade-row propulsor.

It is important to note that an "ideal" duct load for all cases has not been identified. Rather, a duct load for minimum power coefficient for the specific example here has been identified. This example is illustrative of the many different propulsor configurations that must be examined with \( DPLL \) in order to work toward an overall
optimum propulsor. The example here has identified a local optimum propulsor based upon power coefficient for only duct load variation. These results apply only to this particular propulsor configuration and loading.

The duct geometry for the circulation that led to the lowest power coefficient for self-propulsion of the vehicle is shown in Figure 11-7.

11.4 Other possible design techniques with DPLL

Variations in afterbody shape and taper will certainly have some effect on the choice of a propulsor and its powering characteristics. A program was written which will
Figure 11-6: Variation in power coefficient for vehicle self-propulsion vs. a variation in duct circulation. The propulsor is a single blade-row propulsor on an afterbody with small taper. The inflow to the propulsor has no viscous correction to its profile.
Figure 11-7: Duct geometry for minimum self-propulsion power coefficient from Figure 11-6. The propulsor is a single blade-row and the afterbody can be seen to have small taper.
create the set of B-spline vertices needed by DPLL to represent the two-dimensional shape of an input body. A comparison could certainly be carried out by a designer in which a number of different vehicle body shapes are used in DPLL to establish trends of body shape versus some aspect of propulsive performance.

A comparison could also be made of propulsors with varying numbers of stages. A designer could examine trends in some aspect of propulsive performance as the number of blade-rows on a propulsor is increased from one to two to three. Additionally, the ordering of blade-rows in terms of rotating and stationary components could be varied by the designer.

Figure 11-8 is an example to illustrate a DPLL designed geometry for a contra-rotating set of ducted propellers. An example of a geometry with a duct fitted closely to the afterbody is shown in Figure 11-9. The three components of the velocity field (axial, radial, tangential) induced by the geometry shown in Figure 11-9 are shown in Figures 11-10, 11-11, and 11-12, respectively. These contour plots are shown as examples of the type of configurations addressed by DPLL. There are countless additional configurations and duct and blade-row loading distributions that could also have been shown.
Figure 11-8: Geometry of a contra-rotating ducted propulsor on a shaft will small taper.
Figure 11-9: Geometry of a single blade-row propulsor with small duct-body clearance.
Figure 11-10: Axial induced velocity of the geometry shown in Figure 11-9.
Figure 11-11: Radial induced velocity of the geometry shown in Figure 11-9.
Figure 11-12: Tangential induced velocity of the geometry shown in Figure 11-9.
Chapter 12

Future Work on *DPLL*

The purpose of this work was to develop a method for the preliminary design of ducted multiple blade-row propulsors, specifically those on vehicles with highly tapered afterbodies. The input of induced tangential velocity distributions, from which the circulation on each blade-row is calculated, allows the designer control over the induced velocity distributions in subsequent steps of the design of the propulsor.

This technique also allows the comparison of different stern designs for highly tapered vehicles based on power requirements for the propulsion of a given vehicle volume and shape. Also, the effects of different proportions of propulsor thrust carried on the duct can be examined. Finally, the powering characteristics for a given vehicle can be compared for different blade-row arrangements and loading of the blade-rows.

12.1 Vehicle afterbody variations

The vehicle in this method is represented by a number of B-spline vertices. This representation was selected for the smooth and local manner in which the spline
could be affected by a change to one or more vertices. An improvement to the design technique would be a body shape variation routine, possibly with some interior volume specifications for payload size and shape. Variations in the afterbody shape would allow a body optimization iteration with the duct and propulsor design processes already present.

12.2 Duct representation

An improvement to the duct representation could include a thickness model. The propulsor design process would then include duct thickness-induced velocities. Ducts typically are of some appreciable thickness which will have a significant effect on the hydrodynamics of the problem.

12.3 Improved viscous drag models

Viscous drag on blade-rows in DPLL is accounted for with an input viscous drag coefficient. An improved viscous drag model would include the influence of lift coefficient, thickness, and Reynolds number.

Viscous drag on the vehicle body and duct in DPLL is approximated with an input frictional drag coefficient. Frictional resistance comprises a large percentage of the total resistance of the body. An improved resistance calculation for the body and duct would include a calculation of the form drag of both, as well. Information about form drag would help vehicle designers choose between different body shapes in order to obtain minimum overall vehicle resistance. It is expected that form drag of the duct will turn out to be a more significant percentage of the total duct resistance.
than is body form drag of the total body resistance.

12.4 Optimization link to the design process

Rather than design a propulsor based on one or two optimized parameters, DPLL instead considers the entire propulsor and the interactions of its components during a design. DPLL has no optimization techniques, but, in comparison, PLL-4.1 [12] optimizes propeller design primarily through the circulation distribution.

The close proximity of all of the propulsor components and the afterbody will often necessitate significant design changes to multiple components due to an adjustment of a single component. Therefore, it is unreasonable to expect that an optimization of one or two aspects of the propulsor will necessarily lead to overall propulsor optimization in DPLL.

The design process presently works with a fixed body shape, input duct length and camber form, and input distributions of induced tangential velocities on each blade-row. An optimization scheme could be developed around the design process for the development of minimum power configurations of the propulsor. Items which would be included in the optimization include the induced velocity distributions on given blade-rows, the induced velocity distributions between blade-rows, the length of the duct, the load on the duct, the camber form of the duct, and the shape of the afterbody. Optimization techniques could also be applied to determine the appropriate distribution of thrust between the duct and the blade-rows.

A good example of the type of global optimization that is described here regarding DPLL is that developed by Mishima [49] for the optimization of cavitating propeller
blades. Mishima coupled an existing blade design technique to a numerical optimization scheme. In this scheme, the numerical optimization scheme leads to an optimum design based on the results of multiple intermediate blade designs.

12.5 Viscous wake model

*DPLL* includes no viscous adjustment to the inviscid inflow to the duct and the blade-rows. Adjustments to the inviscid inflow profile due to a viscous shear layer should be the next major addition to this work.

Effects of the viscous adjustment of the inflow profile to the propulsor will likely have significant effects on wake convection angles, induced velocity distributions, and blade-row forces. Effects on the duct design will include the duct shape, angle of attack, and forces.
Chapter 13

Incorporation of Analysis

Capabilities into PBD-14/Design

13.1 Background

PBD-14/Design is a vortex-lattice lifting surface computer program capable of the design of single and multiple blade-row open and ducted marine propulsors [3]. The PBD-14 program evolved from many earlier generations of propeller design programs developed at the Marine Hydrodynamics Laboratory at MIT [34, 46].

In the past, propulsor designs created using PBD-14/Design could only be analyzed using separate analysis programs such as PSF-2 or PSF-10 [53, 25]. This analysis process involved time-consuming conversions of PBD-14/Design output files to reflect the format and representations used by each particular analysis program. Additionally, these separate analysis techniques contained only rudimentary treatments of the contraction of streamtube geometry in the vicinity of the propulsor. Thus, a technique by which complex propulsor geometries designed using PBD-14
Design could be quickly and easily analyzed was needed.

In the interest of minimum user time during the design/analysis process, it is most efficient to include the analysis operations as integral parts of the entire PBD-14 code. Although merging the two processes created a longer and more complicated code, the new version avoids the need to move and convert the format of multiple files between two different codes. Since the analysis process works directly off of files generated by the design process, it has also become easier to make necessary coordinated changes to both design and analysis of a propulsor through simple changes to a common input file. Furthermore, changes to logic that is common to both design and analysis can be completed with a change to the single combined design and analysis PBD-14.

The following sections detail the addition of analysis capabilities to PBD-14. The result of this work is a combined design and analysis code which functions either using an input effective inflow or coupled with an axisymmetric viscous flow solver. The flow solver calculates a total inflow from force distributions on the blades. In the coupled case, the effective inflow is extracted from the flow solver total inflow for use in PBD-14. For more information on the difference between the nominal, effective, and total inflow for a propeller, see [54] and [26].

Together with DPLL, the PBD-14/Design and PBD-14/Analysis techniques create a three step process for the design of ducted marine propulsors on highly tapered afterbodies. These steps improve propulsor designs for existing vehicles and aid in the development of propulsors for highly tapered vehicles.
13.2 Induced velocity decomposition and nomenclature

In the case of a coupled design or analysis, information concerning the velocity of the water in the vicinity of the propulsor must be passed back and forth between \textit{PBD-14} and the axisymmetric viscous flow solver. In the case of the \textit{hull} problem, the axisymmetric flow solver creates a circumferential mean "pseudo-total" inflow field in which the propulsor is operating based upon the circumferential mean blade forces calculated in \textit{PBD-14}. In the case of the \textit{blade} problem in \textit{PBD-14}, the design or analysis takes place in an effective inflow which is extracted from the circumferential mean "pseudo-total" inflow output by the flow solver.\textsuperscript{1}

The effective inflow in which the propulsor will operate in reality is not circumferentially constant due to the finite number of blades. Based upon the reasoning in \cite{3} and \cite{42}, however, it is a reasonable approximation to treat the effective inflow as circumferentially mean. Therefore, there is a need to extract an axisymmetric effective inflow from the axisymmetric pseudo-total inflow file. The definition of the effective inflow is the total inflow minus the propulsor induced velocities. It is logical to subtract circumferential mean induced velocities from the pseudo-total inflow to leave the axisymmetric effective inflow. In this work, the abbreviation CMV will be used for the circumferential mean induced velocities of the propulsor.

In order to construct a true total inflow in \textit{PBD-14} for the blade design or analysis problem, the axisymmetric effective inflow must be combined with the non-axisymmetric induced velocities of the propulsor. In this work, the abbreviation LBV

\footnote{It is inconsistent to call the output from the axisymmetric flow solver a \textit{total} inflow as that terminology assumes circumferential variation due to the induced velocities of a finite bladed propulsor. A true total inflow field contains the effective inflow plus the non-circumferential mean induced velocities from the propulsor. The "pseudo-total" inflow produced by the flow solver is axisymmetric.}
will be adopted to describe the non-axisymmetric, or local blade, velocities induced by the propulsor.

This discussion of circumferential mean induced velocities (CMV) and local blade induced velocities (LBV) velocities is carried out at length and in more detail in [3]. That discussion also contains further detail and discussion regarding coupling PBD-14 and the axisymmetric viscous flow solver.
Chapter 14

Design Overview

The purpose of propeller blade design methods like PBD-14/Design is to create a blade shape which produces a designer input distribution of circulation in the radial direction or both radial and chordwise directions. Traditionally, optimized circulation distributions were obtained from the cylindrical lifting line technique PLL-4.1 [12, 8, 13].

14.1 Blade representation

The propulsor's mean camber geometry, in terms of pitch, camber, chordlength, and thickness, as well as the design circulation, are traditionally obtained from parametric studies carried out with the lifting line technique PLL-4.1 [12, 8, 13].\(^1\) Together with designer inputs for blade rake and skew, these geometric quantities for the blades are represented by a fourth order B-spline surface [39, 18] (Figure 14-1).

\(^1\)Though the design input was described here as derived from PLL-4.1, the same description applies to inputs derived in part from the newly developed DPLL. As described in Section 1, the purpose of DPLL is not to replace PLL-4.1 but rather to provide improved preliminary information for design cases of highly tapered designs. Correct preliminary information about these designs is based upon necessary attention to a highly contracted streamtube geometry. DPLL is formulated to account for the strong interactions between the components of a highly tapered
The blade B-spline surface, hub images, and duct images are each represented with user-defined numbers of straight, constant-strength spanwise and free vortices. A vortex lattice blade surface created from the B-spline representation in Figure 14-1 is shown in Figure 14-2.

Associated with each spanwise vortex is a coincident line source to account for blade thickness. Note that an improvement in thickness modeling in *PBD-14* is currently the subject of another research effort. Therefore, thickness is not considered

propulsor.
in the results for the present evaluation of the analysis portion of *PBD-14*, but will be included in the description for completeness.

According to Kelvin's Theorem, Equation 14.1, a vortex line cannot end in the flow. The vortex line must continue infinitely far away or must meet back with itself to form a vortex loop. Any change in the circulation strength along its length must appear as free vorticity shed into the local flow. For a vortex line with continuous, but varying, circulation along its length the circulation is shed into the wake continuously.
along the span of a vortex line according to

\[ \gamma_f(s) = -\frac{d\Gamma}{ds}, \]  

(14.1)

where \( s \) is a local curvilinear coordinate along the vortex.

The continuous distribution of circulation on the blade of a propeller can be represented discretely with vortex segments of finite length and constant circulation along that length, \( \Gamma \). Therefore, the discrete circulation of spanwise vortex \( i \) must be shed from the ends of the vortex as concentrated free vortex lines of strength \( \pm \Gamma_i \). The free vortex lines convect with the local flow and finally into the wake.

### 14.2 Design in an effective inflow

Velocities induced by a vortex lattice propulsor include those due to the circulation of blades, hub image, and duct image spanwise vortices as well as velocities induced by the vorticity shed by each spanwise vortex into the transition and ultimate wakes. In addition are velocities induced by the line sources coincident with each spanwise vortex segment.

During a design, influence functions are calculated for each vortex segment: These influence functions, called local blade velocity (LBV) influence functions, represent the velocity induced by each vortex segment at each control point on the blades. The control points are located adjacent to each spanwise vortex segment on the blade. Each influence function is calculated based on assumed unit vortex strengths. The current process of influence function construction was developed as a foundation for both design and analysis and is detailed in Section 15.1.
Based upon an input radial distribution of circulation on each blade, the circulation strength of each spanwise vortex is assigned: The discrete radial distribution of circulation and thickness are input by the user and are distributed discretely in the chordwise direction based on a NACA $a = 0.8$ mean line distribution.\(^2\)

Using the LBV influence functions for unit strength vortex segments and the assigned strength of each spanwise vortex, the velocities induced by the spanwise and free vorticity of the propulsor are calculated at each control point. To these velocities are added the source (thickness) induced velocities and velocities due to the prescribed propulsor effective inflow and rotation. As described in Figure 14-3, the B-spline representation of the blades is then moved in an iterative manner where the goal is to null, in a least squares sense, the total normal velocity at all of the blade control points. A more detailed account of the entire blade design process can be found in [3].

For the scheme shown in Figure 14-3, the inflow to the propulsor is an effective inflow.\(^3\) The effective inflow to the propulsor in $PBD-14$ is taken as given. This prescribed effective inflow in which the propulsor is designed is typically created based upon a best estimate, model tests, or a computed flow field from some other source. After a particular iteration of a design with $PBD-14$, the output B-spline representation of the blades can be used as input to the next iteration. This process is repeated until the blade shape has converged.

\(^2\)For more on NACA $a = 0.8$ meanlines, see [5].

\(^3\)For more information on the difference between the nominal, effective, and total inflow for a propeller, see [54] and [26]
14.3 Coupling *PBD-14* with RANS

The procedure of using a propeller code coupled with a flow solver has the significant advantage of easily supporting multiple blade row cases. To treat such problems in potential flow alone leads to numerical difficulties as wake sheet singularities from upstream blade-rows approach control points on the downstream blade row. In the coupled method, all of the vortex wake interaction is dealt with by the viscous flow solver so that there are no singular structures and the velocity field is therefore smooth. Thus, *PBD-14* has to deal with only one blade row at a time [3].
When *PBD-14* is to be coupled with a RANS (Reynolds Averaged Navier-Stokes) viscous flow solver, the initial input inflow to the propulsor is typically a calculated or guessed nominal or effective inflow. The first iteration is carried out as shown in Figure 14-3. As part of the output from this first iteration, the circumferential mean forces (CMF) produced by the propeller are calculated. For details of the calculation of the CMF and the reasoning behind their use see [42]. In preparation for the next iteration of the design, the circumferential mean forces are introduced as body forces in the RANS solver to represent the presence of the propulsor. When the RANS code converges on a flow solution in the presence of the propulsor, a second iteration of *PBD-14* is started with this new pseudo-total inflow and the first iteration’s output blade B-spline control polygon. During and after the second iteration of *PBD-14* with RANS, the input inflow file will contain a certain amount of circumferential mean velocity caused by the RANS inclusion of the CMF.

### 14.4 Effective inflow in *PBD-14/Design*

Whether used for design in an input effective inflow or used coupled with a viscous flow solver, *PBD-14/Design* needs an effective inflow in which to carry out the design. In the case of an input effective inflow the design takes place as shown in Figure 14-3. When coupled with the viscous flow solver, however, *PBD-14/Design* is provided with an input inflow velocity field that also contains the circumferential mean velocities from the inclusion of the previous iteration’s CMF in the viscous flow solver. The effective inflow must be extracted from this input inflow velocity field. As diagrammed in Figure 14-4, the presence of the circumferential mean velocities in the
inflow field due to the previous iteration’s CMF is roughly accounted for during the current iteration by subtracting out the circumferential mean portion of the influence functions (CMV) from the LBV influence functions. This is described in Figure 14-4.

Figure 14-4: Flowchart of *PBD-14/Design* using RANS inflow iterations.

The velocities at each control point calculated during each iteration will thus be composed of the viscous flow solver inflow field, the velocities due to rotation of the propeller, and the difference between the current LBV and CMV induced velocities. Though the circumferential mean velocities for the present iteration are not exactly equal to the previous iteration’s CMF induced circumferential mean velocities, the
approximation is close enough to allow future iterations through $PBD-14$ and RANS to converge on circumferential mean induced velocities that are essentially identical from one iteration to the next. The convergence of the design process generally occurs in 4 to 6 iterations between $PBD-14$ and RANS [16], as shown in Figure 14-5.

Convergence of the propeller forces typically occurs earlier than the actual blade shape convergence. Therefore, blade shape convergence is typically used as an indicator of general convergence of the design. Figure 14-5 shows the convergence of the blade shape for a single open propeller on a notional afterbody similar to the geometry shown in Figure 17-10. Convergence is shown in terms of the root mean square of the adjustment to all of the blade control points at each iteration. The design is coupled to the RANS viscous flow solver after every seventh blade iteration in $PBD-14$. 

Figure 14-5: *PBD-14* coupled design convergence of blade shape iterations.
Chapter 15

Analysis Background

15.1 Horseshoe influence function theory

In both the design and analysis processes, the velocities induced by blade, hub, and duct singularities and wakes are represented in terms of "horseshoe influence functions". Each horseshoe represents the three dimensions of velocity induced by a given blade spanwise vortex, of unit strength, on a given control point. Because the location and orientation of each vortex relative to each control point is known during a design from the initial input geometry, induced velocities can be calculated from Biot-Savart’s Law [43]. The subroutine VORSEG[41] employs numerical integration approximations of Biot-Savart’s Law and creates an influence function of the summed effects of a vortex segment and the identical segments on the remaining blades of the propulsor at a given control point. Blade thickness, represented by a known (input) distribution of sources over the blades is not accounted for in the horseshoe influence functions.

Though the process of the construction of horseshoes is fairly simple, it is heavily
dependent on book-keeping and index switching: Kelvin’s Theorem (Equation 14.1) dictates that the induced velocity effect of each spanwise vortex must also include the effect of the vorticity shed from each end of that element. Thus, each horseshoe influence function also includes the induced velocities due to the chordwise trailing vortex elements that trail off of the spanwise vortex segment. The chordwise elements are convected with the flow passing the blade to eventually form the transition and ultimate wakes.

The construction of horseshoes results in horseshoe influence functions that take into account the induced velocity effect of each blade spanwise vortex segment on each control point on the blade, including the influence of that vortex’s images in the hub and duct and the identical blade spanwise segments, images, and trailers on the remaining blades. As there are equal numbers of blade spanwise vortices and control points, for even modest blade discretization and imaging the total number of horseshoe influence functions and associated computations grows rapidly.

15.2 Hub and duct images

An additional level of complexity is added when images representing the duct and hub are added to the problem. By definition, these images are equal and opposite representations of the spanwise vortices on the blade: A blade vortex is imaged by a duct or hub spanwise vortex of equal strength. The influence of each duct and hub image spanwise vortex is added to the horseshoe for the blade spanwise vortex that is being imaged. As with the blade spanwise vortices, the effect of an image spanwise hub or duct vortex includes the effect of all of the trailing vorticity and its transition
and ultimate wakes.

15.3 Horseshoe influence function verification

The proper construction of the horseshoe influence functions was verified during the development of the analysis sections of *PBD-14*. The influence functions were multiplied by an input distribution of spanwise vortex strengths. These induced velocities determined from the new analysis routine agreed to within single precision tolerances with those calculated by a velocity calculation technique which existed in the design-only version of the code [43, 22].

The original design method calculated induced velocities at blade control points without the need to total and save influence functions as it progressed. Thus, the calculation of velocities for a given control point proceeded simply by calculating the induced velocity on the control point due to a given vortex segment, multiplying the influence function by the segment’s known circulation, and then proceeding to the next vortex segment. The induced velocity was totaled as the program proceeded and accounted for all vortex segments. On the other hand, the new analysis method needs large arrays in computer memory in which to store the induced velocity influence function for every vortex segment’s influence on every control point. Because the analysis technique needs access to the entire array of horseshoe influence functions, large memory requirements are unfortunately unavoidable; in any case, however, these requirements are well within the capacity of most current workstations and personal computers.

The stored arrays are not necessary for design, so design could be carried out by
the original technique without the memory and arrays necessary for analysis. In an effort to make PBD-14 more modular so that future improvements or changes to the induced velocity logic are easy to implement, the older design method was eliminated in favor of the new technique which sums and saves influence function arrays. Any future changes to the way in which induced velocities are calculated can be effected by a change to a single set of influence function subroutines which construct influence functions for either design or analysis.
Chapter 16

Solving for Blade Circulation

16.1 Horseshoe influence function matrix

During a design, the blade is assigned prescribed spanwise vortex strengths based upon user input circulation distributions. The blade shape was then adjusted iteratively until the summed effect of all singularities, the inflow, and the rotationally induced velocities at each control point on the blade resulted in minimum normal total velocity, in a least squares sense, over the entire blade surface.

In analysis, the construction of the horseshoe influence functions leads to a matrix of influence functions that represent the effect of each spanwise vortex (and all trailing vorticity and all images) on each control point for the given propulsor geometry to be analyzed. From this matrix, and the known values of the normal effective inflow at each control point on the blade, the analysis portion of PBD-14 must solve for the strength of each of the spanwise vortices so that, when combined with the rest of the flow, the vortex induced velocities will result in zero normal total flow velocity at each control point.
The horseshoe influence matrix is sent to a solver within the program which determines the appropriate strength of each blade spanwise vortex in order to satisfy the zero normal flow condition. The final adjustment to the calculation comes from the effect of thickness (represented by sources) of the blade, duct and hub. If thickness were present in an analysis calculation, these source induced velocities would be added onto the effective inflow which is used as the right hand side of the matrix solving equations. The source strengths are calculated from a known blade thickness distribution, the known effective inflow, and propeller rotation.

16.2 Solved circulation

The design process is incapable of completely nulling normal velocity at every control point simultaneously, due to the B-spline representation of the blade. This type of blade representation must balance surface fairness with the accurate positioning of the panels representing the blade for the assigned circulation distribution. Though it does a very good job of balancing the normal velocity and fairness of the surface, there are always residual normal velocities on the designed blade. An example of residual normal velocities are shown in Figure 16-1 as velocities normalized on the free stream velocity.

When the analysis routine solves for spanwise vortex strengths that will null normal velocities at each control point, the vortex strength distribution will be at least slightly different from the circulation distribution input to the design. One benefit of an analysis routine is the capability to determine how closely the design technique has come to creating a blade with the prescribed design circulation distribution.
Figure 16-1: Contour plot of PBD-14/Design residual normal velocities on the blade.
Chapter 17

Verification of Analysis in an Effective Inflow

As with design, analysis of a propulsor in \textit{PBD-14} can be carried out in a prescribed effective inflow or can be coupled with a viscous flow solver to converge on a blade design and a total inflow in which that propulsor will operate. This section covers a few design and analysis examples to help verify the analysis technique.

Figure 17-1 shows the program flow for an analysis in a prescribed effective inflow. Given the propulsor geometry and prescribed effective inflow, a single pass through \textit{PBD-14} analysis results in the solved circulation distribution over the blades of the propeller and the resultant blade forces. Additional output files and information are also created for both design and analysis in \textit{PBD-14} [3].

The first technique for verifying the analysis routines was to design a propeller and then analyze it in \textit{PBD-14} in the same effective inflow field in which it was designed. A sample propeller (Figure 17-2) was designed and then analyzed in a
Figure 17-1: Flowchart of PBD-14 Analysis in a prescribed effective inflow.
uniform effective inflow. The design was carried out without any hub or duct images and with a circulation distribution which goes to zero at the hub and tip. Likewise, the analysis was carried out without hub and duct images. The radial distribution of circulation to which the propeller was designed (Figure 17-3) compares quite well with the actual distribution analyzed from the designed geometry. The design and analyzed circulation distributions in both the spanwise and chordwise directions over the blade (Figure 17-4) also show good design/analysis agreement.

A comparison of the designed and analyzed radial distribution of circulation (Figure 17-6) shows good agreement for a propeller (Figure 17-5) designed and analyzed using hub images. The design circulation distribution has finite loading at the hub and loading that goes to zero at the tip of the blade. The hub-loaded circulation results also show good design/analysis agreement in both the chordwise and spanwise directions (Figure 17-7).

Hub and duct images were both used in PBD-14 for an additional design/analysis comparison of the propeller shown in Figure 17-5. The radial circulation distributions (Figure 17-8) show that this propeller carries load at both the hub and the tip of the blades. Figure 17-8 shows good agreement between the design and analysis radial distributions of circulation while Figure 17-9 shows good agreement between the distributions over the entire blade surface.

As PBD-14 is capable of representing generalized streamtube geometries, additional design/analysis verifications were performed on a propulsor fitted to a tapered centerbody (Figure 17-10). This geometry was designed and then analyzed in a prescribed effective inflow that varied with radius. Comparisons of the design and
Figure 17-2: Five blade sample propeller for design/analysis verification.

Figure 17-3: Design vs. analyzed circulation by radius, no hub or duct images.
Figure 17-4: Design vs. analyzed circulation over entire blade, no hub or duct images.

Figure 17-5: Four blade sample propeller for design/analysis verification.
Figure 17-6: Design vs. analyzed circulation by radius, hub images used.

Figure 17-7: Design vs. analyzed circulation over entire blade, hub images used.
Figure 17-8: Design vs. analyzed circulation by radius, hub and duct images used.

Figure 17-9: Design vs. analyzed circulation over entire blade, hub and duct images used.
analysis circulation distributions are shown in Figures 17-11 and 17-12. The analyzed circulation distribution matches the design distribution well both in magnitude and form.
Figure 17-10: Five blade sample propeller for design/analysis verification: tapered body.

Figure 17-11: Design vs. analyzed circulation by radius, tapered body, hub images used.
Figure 17-12: Design vs. analyzed circulation over entire blade, tapered body, hub images used.
Chapter 18

Coupling $PBD-14$/Analysis with RANS

As with analysis in a prescribed effective inflow, the desired final form of the circulation solution equation for analysis in $PBD-14$ is:

$$\begin{bmatrix} LBV \end{bmatrix} \begin{bmatrix} \Gamma \end{bmatrix} = - \begin{bmatrix} VEF \end{bmatrix}$$

where:

$$\begin{bmatrix} LBV \end{bmatrix} = \text{Matrix of Influence Functions}$$

of Total Induced Velocity

Normal to Blades

$$\begin{bmatrix} \Gamma \end{bmatrix} = \text{Strength of Blade Horseshoe}$$

Spanwise Vortex Elements
\[ [ VEF ] = \text{Effective Inflow Normal to Blade} \]

The flow-field output from RANS contains the effective inflow plus circumferential mean induced velocities due to blade CMF forces (See Section 14.3). Therefore, for analysis coupled with RANS, the circulation solution equation instead must become:

\[
[ LBV - CMV ] [ \Gamma ] = - [ VEF + RMV ]
\] (A)

where:

\[ [ CMV ] = \text{Circumferential Mean Velocity} \]

Normal to Blade Induced by Blade

Horseshoe Influence Functions

\[ [ RMV ] = \text{Circumferential Mean Induced Velocities} \]

Normal to Blade from RANS Solution Based on Blade Body Forces from the Blade Solution.

It is important to note that RANS provides the “total” velocity \([ VEF + RMV ]\); RANS cannot distinguish between the two components.
The next step is to formulate the iterative Analysis/RANS scheme. The most obvious approach, starting from Equation $A$, is to simply manipulate the matrices and write

$$[\Gamma]^i = - \left[ \overline{LBV} - \overline{CMV} \right]^{-1} \left[ VEF + RMV \right]^{i-1},$$

where $i$ is the iteration number. The matrix $\left[ VEF + RMV \right]^{i-1}$ is the RANS flow field containing the combined effective inflow and circumferential mean induced velocities due to blade forces from the previous iteration of $PBD-14$ analysis. The matrix $\left[ \overline{LBV} - \overline{CMV} \right]^{-1}$ is the inverse matrix of the difference between the local and the circumferential mean velocity influence functions.

Unfortunately, this iterative scheme is unstable and fails to converge. The entries in the combined matrix $\left[ \overline{LBV} - \overline{CMV} \right]$ are very often finite values but often the difference between the $LBV$ and $CMV$ influence functions for a given control point due to a given spanwise vortex can be very close to zero. It is also possible for these influence functions to be small negative numbers.

The danger with this type of influence function matrix is that the small influence functions can call for unrealistically large solved spanwise vortex strengths in some locations so that the resultant induced velocities are of the necessary magnitude to balance the inflow at each control point. The small negative influence functions can likewise result in unrealistic circulation strengths. The resultant circulation distribution on the blades of the propeller is used for the circumferential mean force calculation. The viscous flow solver is provided with these polluted blade forces and in this way the next $PBD-14$ run is further contaminated by way of the inflow,
\[ VEF + RMV \]. The iterative scheme diverges with increasing number of iterations between PBD-14 and the RANS viscous flow solver.

Therefore, Equation A is reformulated by moving \[ CMV \] \[ \Gamma \] to the right hand side of the equation:

\[
\begin{bmatrix}
  LBV
\end{bmatrix}
\begin{bmatrix}
  \Gamma
\end{bmatrix} = -( \begin{bmatrix}
  VEF + RMV
\end{bmatrix} - \begin{bmatrix}
  CMV
\end{bmatrix} \begin{bmatrix}
  \Gamma
\end{bmatrix} )
\]

The term \[ CMV \] \[ \Gamma \] is the circulation on the blade multiplied by the circumferential mean horseshoe influence functions. This is the present iteration's circumferential mean induced velocities on the blade. The equation shows the solution of blade circulation to be a function of itself; it can only be calculated if it is already known. Therefore, this equation was rewritten into a stable iterative scheme where the solution to the circulation depends instead on the previous PBD-14 iteration’s output circumferential mean induced velocities (CMV) and the circumferential mean velocities contained in the RANS flow field result (RMV) produced from the previous iteration’s blade forces.

\[
\begin{bmatrix}
  \Gamma
\end{bmatrix}^i = -[LBV]^{-1} ( VEF + RMV)^{i-1} - [CMV] \begin{bmatrix}
  \Gamma
\end{bmatrix}^{i-1}
\]

For the first iteration of coupled PBD-14 analysis, the initial input inflow field (RMV) can be an estimated or calculated nominal, effective, or total inflow field. If initial values for the CMV field are not available for this geometry at similar operating conditions, the first iteration of the analysis process must follow the outline in Figure 17-1. After producing a CMV file, the second and all subsequent analysis iterations can then proceed as shown in Figure 18-1.
Figure 18-1: Flowchart for PBD-14/Analysis Using RANS Inflow Iterations.

Unlike analysis with a prescribed effective inflow, analysis coupled with RANS cannot go directly to the solution of the blade circulation using the input flow field. The input RANS inflow file necessarily contains some form of an effective inflow field plus velocities induced in the RANS viscous flow solver by the circumferential mean forces (CMF) produced in PBD-14 by the propulsor. As stated earlier, RANS cannot distinguish between the effective inflow and induced velocity components of the velocity field it produces. Therefore, PBD-14 must calculate the effective velocities at the control points on the blade. This is performed by subtracting the control points’
circumferential mean velocities, obtained from the previous iteration of \textit{PBD-14} analysis, from the input RANS velocity field. These resultant velocities are effective inflow velocities at the control points on the blade. Once this subtraction is complete, \textit{PBD-14} can then proceed with the solution of the blade circulation in the same manner as if it had been provided with an effective inflow in the first place.

Though the previous iteration’s CMV are by no means expected to be the same as those which will be calculated during the present iteration, the previous CMV are used to provide the solver with an effective inflow from which the current circulation distribution can be obtained. This circulation distribution is then used to produce a CMV for the next \textit{PBD-14} iteration, a CMF for the next RANS solution of a new input inflow field, and current propulsor forces from which the designer can monitor convergence of the method. Convergence of the CMV, and thus propulsor forces, typically occurs in 4 to 6 iterations between \textit{PBD-14} analysis and RANS [16]. An example of this convergence is shown in Figure 18-2 for DTRC propeller 4577 on an afterbody with small taper.
Figure 18-2: Convergence of propeller forces versus number of PBD-14/RANS iterations for DTRC propeller 4577 with coupled PBD-14/Analysis.
Chapter 19

Verification of Coupled Design and Analysis

The propeller geometry shown in Figure 19-1 was designed using the coupled PBD-14/RANS design method in a uniform effective inflow. Following that design, the coupled PBD-14/RANS analysis method was begun with the final CMV file from the design. The analysis was carried to convergence of the total propeller forces. Shown in Figure 19-2 is a comparison between the design radial distribution of circulation and that recovered by the analysis. Similarly, Figure 19-3 shows the circulation comparison over the entire blade surface. In both cases there is good agreement between the design and analysis results for this coupled example. Experience with similar comparisons has shown like agreement for different propeller geometries.
Figure 19-1: Propeller geometry for coupled design/analysis verification: straight shaft.
Figure 19-2: Radial distribution of circulation for coupled $PBD-14$ /Design vs. coupled $PBD-14$ /Analysis, no hub or duct images.
Figure 19-3: Circulation over entire blade for coupled PBD-14 /Design vs. coupled PBD-14 /Analysis, no hub or duct images.
Chapter 20

Comparison of Coupled

PBD-14 / Analysis

with Experimental Results

Validation of the newly developed PBD-14 / Analysis program versus experiment is discussed in this section. Though PBD-14 / Analysis is capable of modeling the conical streamtubes of highly tapered afterbodies, this validation is carried out for a propeller on a straight shaft. There are no experimental results currently available for circulation and force measurements of a propulsor on a highly tapered afterbody.

Experimental results for the radial distribution of circulation and propeller forces were available for Propeller 4119 operating in uniform inflow as obtained by Jessup [30]. The experimental circulation distributions are based upon tangential velocities measured in the wake of the propeller. The experimental circulation distribution measurements, as well as propeller force measurements, were performed at the design
advance coefficient of \( J = 0.833 \). Propeller 4119 was designed by Denny [15], at the David Taylor Research Center, for uniform inflow.

Figure 20-1: Propeller 4119, used for coupled \textit{PBD-14} verification

A coupled \textit{PBD-14} /Analysis was carried out using Propeller 4119. An uncoupled \textit{PBD-14} /Analysis in a prescribed effective wake was also carried out. The uncoupled analysis was performed with wake convection velocities at the plane of the propeller as calculated from \textit{PSF-3} [36]. The calculations performed in this section, with all of the analysis codes, were performed with a coefficient of viscous drag of 0.0035, unless otherwise stated.
Diameter, \( D = 1.00 \text{ ft.} \ (0.305\text{m}) \)
Rotation: Right Hand
Number of Blades: 3
Hub-Diameter Ratio: 0.20
Skew \( \varphi \), 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>( i_T/D )</th>
<th>( i_M/C )</th>
<th>( i_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>0.0000</td>
<td>1.075</td>
<td>0</td>
<td>0</td>
<td>0.03160</td>
<td>0.01175</td>
</tr>
</tbody>
</table>

Table 20.1: Geometry of DTRC Propeller 4119.

As the improvement of the thickness induced velocity logic in \( PBD-14 \) is currently the subject of a separate research effort, thickness was not included in the \( PBD-14 \) computations. Experimental results for 4119 contain the effects of blade thickness. Therefore it is expected that the circulation distribution results shown in Figure 20-2 show differences between the \( PBD-14 \) computations and the experiment. \(^1\) The circulation distribution differences are reflected also in the force comparisons in Table 20.2.

In order to reconcile the experimental and \( PBD-14 \) results, a comparison was performed between the experiment and results from an existing analysis panel code, \( PSF-\)

\(^1\)The small jump in circulation around \( r/R = 0.2 \) for the coupled \( PBD-14 \) results is Figures 20-2 and 20-4 is due to a suboptimal interpolation scheme during the transfer of the the inflow file from the flow solver to the \( PBD-14 \) analysis method. In practice, the circulation at the root of the propeller (\( r/R = 0.2 \), in this case) would have been determined by some form of interpolation from the adjacent radii.
10 [24]. PSF-10 calculates thickness induced velocities for the propeller. Figure 20-3 compares the experimental results with the PSF-10 results, including thickness induced velocities, for Propeller 4119. The PSF-10 circulation distribution compares well with the experimental results. Likewise, there is very good force agreement between the computed and measured forces (Table 20.3).

Next, PSF-10 was run using 50% and 150% of the true blade thickness so that PSF-10 results could be plotted for a variety of thicknesses (Figure 20-4). The resulting circulation distributions are shown along with the coupled no-thickness PBD-14
<table>
<thead>
<tr>
<th></th>
<th>$K_T$</th>
<th>$10^*KQ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>0.146</td>
<td>0.280</td>
</tr>
<tr>
<td>$PBD-14$ coupled</td>
<td>0.1655</td>
<td>0.3038</td>
</tr>
<tr>
<td>$PBD-14$ coupled difference</td>
<td>+13.4%</td>
<td>+8.5%</td>
</tr>
<tr>
<td>$PBD-14$ uncoupled</td>
<td>0.1676</td>
<td>0.3079</td>
</tr>
<tr>
<td>$PBD-14$ uncoupled difference</td>
<td>14.8%</td>
<td>+10.0%</td>
</tr>
</tbody>
</table>

Table 20.2: Propeller 4119 forces from $PBD-14$ and experiment

<table>
<thead>
<tr>
<th></th>
<th>$K_T$</th>
<th>$10^*KQ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>0.146</td>
<td>0.280</td>
</tr>
<tr>
<td>$PSF-10$ (100% thick)</td>
<td>0.1470</td>
<td>0.2724</td>
</tr>
<tr>
<td>$PSF-10$ difference</td>
<td>+0.7%</td>
<td>-2.7%</td>
</tr>
</tbody>
</table>

Table 20.3: Propeller 4119 forces from $PSF-10$ and experiment

results in Figure 20-4. A clear trend in $PSF-10$ circulation results approaching that of $PBD-14$ is evident as $PSF-10$ approaches a limit of zero thickness. Finally the inviscid forces produced by $PSF-10$ were extrapolated to zero thickness for comparison with inviscid $PBD-14$ in Table 20.4. There is very good agreement between the $PBD-14$ no-thickness results and the $PSF-10$ no-thickness results.

Based upon the good performance of $PSF-10$ at true (100%) thickness compared to experiment, the very similar circulation and force performance of no-thickness $PBD-14$ with no-thickness $PSF-10$ indicates that the coupled $PBD-14$ analysis method is performing properly. The addition of an improved thickness-induced velocity scheme will further strengthen $PBD-14$. The good performance of $PBD-14$ /Analysis in this validation is encouraging, supporting the hope of equal success in validations against possible future experiments of propulsors on highly tapered afterbodies.
Figure 20-3: *PSF-10* and experimental results for Propeller 4119

<table>
<thead>
<tr>
<th></th>
<th>(K_T)</th>
<th>(10^*KQ)</th>
</tr>
</thead>
<tbody>
<tr>
<td><em>PSF-10</em> (150% thick)</td>
<td>0.1462</td>
<td>0.2297</td>
</tr>
<tr>
<td><em>PSF-10</em> (100% thick)</td>
<td>0.1543</td>
<td>0.2495</td>
</tr>
<tr>
<td><em>PSF-10</em> (50% thick)</td>
<td>0.1620</td>
<td>0.2662</td>
</tr>
<tr>
<td><em>PSF-10</em> (0% thick)</td>
<td>0.1693</td>
<td>0.02798</td>
</tr>
<tr>
<td><em>PBD-14</em> (0% thick)</td>
<td>0.1686</td>
<td>0.02791</td>
</tr>
<tr>
<td><em>PBD-14</em> difference from <em>PSF-10</em></td>
<td>-0.4%</td>
<td>-0.25%</td>
</tr>
</tbody>
</table>

Table 20.4: Propeller 4119 inviscid forces from *PSF-10* and *PBD-14*
Figure 20-4: PSF-10 results with thickness variations for Propeller 4119
Chapter 21

Future work on $PBD-14$

21.1 Blade thickness and viscous modeling

An improvement of modeling of blade thickness and blade viscous effects is currently the subject of another research effort by Black [4]. The current viscous scheme involves an input coefficient of drag which is applied stripwise on the blade surface.

21.2 Tip-gap flows

A tip gap model is currently the work of McHugh [48]. This work involves improvements to the hub and duct images by representing the hub and duct with vortex lattices on which the kinematic boundary condition can be satisfied in conjunction with that on the blade surfaces. Flow through the tip region will be approximated with a tip row of panels upon which the kinematic boundary condition can be satisfied using a semi-empirical orifice flow model. As explained in the current work, the presence of the hub and duct is presently represented with spanwise and trailing
image vortices.
Chapter 22

Conclusions

A design procedure tailored to the design of ducted propulsors on highly tapered afterbodies was developed. This process includes three steps: a new preliminary design tool, an existing blade shape design technique, and a new blade shape analysis technique. This three step procedure improves propulsor designs for existing vehicles and aids in the development of propulsors for notional highly tapered vehicles.

22.1 Derivation of new non-dimensional coefficients

Consideration of highly tapered geometries leads to difficulties with traditional definitions for propeller non-dimensional coefficients. As detailed in Section 9, the difficulties center around the measure of diameter for highly tapered propulsors. Due to finite blade-row tip chordlengths that vary in radius and hubs whose diameters vary from blade leading edge to trailing edge, the definition of blade-row diameter is ill-defined. The decision was made to abandon traditional definitions which are normalized by propeller diameter or radius. Instead, the few essential coefficient
definitions were recreated based instead on maximum vehicle diameter.

There is a potentially large disparity between the unpropelled and the propelled resistance of highly tapered vehicles. This is due to the tendency of the propulsor to reduce or eliminate separated flow which would have occurred in the absence of the propulsor. For this reason, the use of traditional definitions of efficiency, which are defined using unpropelled resistance, could be dangerous. In fact, the definition of an efficiency for highly tapered vehicles was avoided completely in this work. A primary purpose of improving a propulsor design is to minimize the power necessary to propel the vehicle given a set of operating conditions. The comparison of different propulsor configurations can therefore be made by way of the newly normalized power coefficient, based on the power consumed by the propulsor.

22.2 The new design procedure

Shown in Figure 22-1 is the traditional design procedure for marine propulsors. Preliminary design typically depends upon cylindrical lifting line techniques and streamline curvature methods. Blade shape design is carried out with PBD-14/Design. Blade shape analysis can be carried out in various analysis programs.

Shown in Figure 22-2 is the newly developed design procedure for marine propulsors on highly tapered afterbodies. The newly developed preliminary design technique is DPLL. Blade shape design and analysis are performed with the combined design and analysis program made up of PBD-14/Design and the newly developed PBD-14/Analysis.
Figure 22-1: Schematic of the traditional propulsor design process

Figure 22-2: Schematic of the new design process for propulsors on highly tapered vehicles.
22.2.1 Preliminary design

The newly developed preliminary design tool, DPLL, is a lifting line technique for multiple blade-row ducted propulsors on tapered afterbodies. Rather than optimize pieces of the propulsor individually, DPLL models all of the components of a highly tapered propulsor, including the afterbody surface. Due to the close proximity of the duct, blade-rows, and afterbody in this type of propulsor, it is important for the designer to have a tool that models the interaction of these components. Change to one component can necessitate significant associated changes to the other components. For this reason it is unreasonable to expect that an optimization of one or two aspects of the propulsor will necessarily lead to overall propulsor optimization. A process for the identification of superior propulsor configurations must consider the duct, blade-rows, and afterbody as one integrated unit. Modeling of contracted streamtube geometries and a procedure for a duct mean-camber design are included in DPLL. The effect of variations in vehicle afterbody taper can also be examined.

The ideal circulation distribution on multiple blade-row ducted propulsors is generally known. Rather than optimize the circulation distribution, DPLL allows a user specification of the form of the induced tangential velocity on each blade-row of the propulsor. The circulation distribution on each blade-row is then generated from the tangential induced velocity specification. In this manner, the designer can work to identify an induced tangential velocity distribution that is acceptable for later steps in the design process while also matching the desired circulation distribution.

Prior to the development of DPLL, the preliminary phase of a design often depended upon calculations from cylindrical lifting line methods [12] or streamline cur-
vature methods [57, 47]. As described in Section 1, cylindrical lifting line methods assume that the propulsor wake is purely helical, an inappropriate assumption for propulsors on highly tapered afterbodies with resultant contracted streamtube geometries. Streamline curvature methods account for contracted streamtube geometries, but often do not model the flow on the outside of the duct. Therefore, these methods are inappropriate as they do not model the presence of the duct adequately.

DPLL is intended for large parametric studies where basic accuracy and speed are the main goals. The primary inputs to this theory are a centerbody geometry, the axial location and extent of the duct, the magnitude of duct load, the number of blade-rows, and the form of the self-induced tangential velocity on each one of the lifting line blade-rows. The important results of a given design with this theory are the duct shape and orientation, the distribution of circulation on each of the lifting line blade-rows, and a measure of the propulsor power consumption.

22.2.2 Blade design

Blade design is performed with the pre-existing PBD-14 /Design method, a vortex-lattice lifting surface design tool that carries out designs in an effective inflow. The effective inflow can either be specified by the designer or calculated within PBD-14 /Design from the output of a flow solver coupled to PBD-14 /Design. PBD-14 /Design accounts for the contracted trailing vortex geometry of propulsors on highly tapered afterbodies. The model also includes the induced velocities of the hub and duct.
22.2.3 Blade analysis

The newly developed blade shape analysis technique is $PBD-14$ /Analysis, which has been embedded within pre-existing code to create an overall $PBD-14$ design/analysis program. The combined design/analysis format allows for quick and easy changes from design to analysis and back with simple changes to only one input file. The form of inputs and outputs in $PBD-14$ is identical between design and analysis.

Prior to the current work, analysis was typically carried out in separate analysis codes. As described in Section 1, these analysis techniques had shortcomings in their treatment of contracted streamtube geometries. Also, it was necessary to convert the output files from $PBD-14$ /Design into the very different input file format of the separate analysis codes.

$PBD-14$ /Analysis analyzes blade designs in an effective inflow, which can be designer specified or internally calculated from the output of an optional coupled viscous flow solver. $PBD-14$ /Analysis accounts for the contracted trailing vortex geometry of propulsors on highly tapered afterbodies, and also includes the induced velocities of the hub and duct.

Verification of $PBD-14$ /Analysis against $PBD-14$ /Design showed good agreement between the design and analysis methods. The addition of an improved thickness-induced velocity scheme will further strengthen $PBD-14$ for both design and analysis. The good performance of $PBD-14$ /Analysis in a validation against experimental results for a propeller on a straight shaft is encouraging. This validation supports the hope of equal success in validations against possible future experiments of propulsors on highly tapered afterbodies.
22.3 Summary

From preliminary design to analysis of the final design, the newly developed design procedure is tailored to the development of propulsors for highly tapered afterbodies. Ideally, this procedure will allow the development of propulsor designs for highly tapered afterbody vehicles while maintaining powering characteristics similar to current vehicles. Additionally, the streamtube contraction treatment within DPLL may improve current propulsors that were designed with cylindrical streamtube assumptions but are fitted on vehicles with small but finite afterbody taper.

Advantages of a propulsor design that accommodates high vehicle afterbody taper include increased capacity of the body in the region where most of the propulsion systems are located. The increased volume aft could also allow the vehicle a greater energy storage capacity, payload volume and weight, or improved handling and station keeping characteristics. The change in the distribution of submerged volume may also allow designers additional freedom for improvements to the design of the remainder of the vehicle. Effects of improvements could include decreased vehicle size, weight, length, and cost.
Appendix A

Cubic Spline Representation of
Propeller Induced Velocities

Cubic spline representations of a set of data which includes some form of “jump” can often misrepresent the data. Similar to the data represented by the square symbols in Figure A-1, there is a jump in tangential velocity as one passes axially (increasing axial coordinate in the figure) through the plane of a propeller. The smooth curve shown is a cubic spline representation of that jump. The spline representation clearly alters the nature of the input data, which is zero upstream and of unit value downstream of the propeller plane.

The approximation of jumps in data by cubic splines becomes even more suspect when one wants to use the cubic coefficients to extrapolate values beyond where the input data was available. Figure A-2 shows the same input data used in Figure A-1, but now the cubic spline is evaluated upstream and downstream beyond the range of the input velocity data. Not only is the cubic spline a poor representation of the
input data, but the approximation becomes very bad very fast with small increases in distance from the input range.

![Cubic spline representation of a discrete velocity jump](image)

**Figure A-1:** Cubic spline representation of a discrete velocity jump

Additionally, the value of the cubic spline when extrapolated beyond the input data range is somewhat arbitrary based upon the number of data points originally used to build the cubic coefficients. In Figure A-2, there are four input data points downstream of the jump location and the extrapolated values clearly head toward large negative values. If there had been an additional input data point at, say, $X=8.0$ and $\text{Tangential Velocity}=1.0$, the extrapolated values downstream of the input data range would now head toward large positive values!

When the input data contains a more smoothly varying velocity field, the cubic spline representation, as shown in Figure A-3, is actually quite good. The extrapolated values for this case also appear to be quite good. A warning is still appropriate
regarding extrapolated values, as the author, while splining slight variations of the input data field shown in Figure A-3, easily found additional smooth data fields which also had good cubic spline representations within the range of the input data but exhibited unpredictable and sometimes poor extrapolated values. The lesson learned was to extrapolate only from smooth input data and extrapolate only for very short distances beyond the input data range.
Figure A-3: Cubic spline representation of a smooth velocity variation
Bibliography

[1] Pennsylvania State University
   Intercollege Research Programs and Facilities
   APPLIED RESEARCH LABORATORY
   Post Office Box 30
   State College, PA 16804.


[5] T. E. Brockett. Minimum pressure envelopes for modified naca-66 sections with naca a=0.8 camber and buships type i and type ii sections. Report 1780,


[41] J. E. Kerwin. Subroutine VORtex SEGment: Vorseg, Multiple years and versions. Vorseg exists in many forms in many MIT codes.


