Accelerating Conceptual Design Analysis of Marine Vehicles through Deep Learning

Matthew C. Jones

Dissertation submitted to the Faculty of the Virginia Polytechnic Institute and State University in partial fulfillment of the requirements for the degree of

Doctor of Philosophy
in
Aerospace Engineering

Eric G. Paterson, Chair
William J. Devenport
Christopher J. Roy
Jonathan S. Pitt

April 15, 2019
Blacksburg, Virginia

Keywords: near wake, machine learning, deep learning, adversarial network, OpenFOAM
Copyright 2019, Matthew C. Jones
Evaluation of the flow field imparted by a marine vehicle reveals the underlying efficiency and performance. However, the relationship between precise design features and their impact on the flow field is not well characterized. The goal of this work is first, to investigate the thermally–stratified near field of a self–propelled marine vehicle to identify the significance of propulsion and hull–form design decisions, and second, to develop a functional mapping between an arbitrary vehicle design and its associated flow field to accelerate the design analysis process. The unsteady Reynolds–Averaged Navier–Stokes equations are solved to compute near–field wake profiles, showing good agreement to experimental data and providing a balance between simulation fidelity and numerical cost, given the database of cases considered. Machine learning through convolutional networks is employed to discover the relationship between vehicle geometries and their associated flow fields with two distinct deep–learning networks. The first network directly maps explicitly–specified geometric design parameters to their corresponding flow fields. The second network considers the vehicle geometries themselves as tensors of geometric volume fractions to implicitly–learn the underlying parameter space. Once trained, both networks effectively generate realistic flow fields, accelerating the design analysis from a process that takes days to one that takes a fraction of a second. The implicit–parameter network successfully learns the underlying parameter space for geometries within the scope of the training data, showing comparable performance to the explicit–parameter network. With additions to the size and variability of the training database, this network has the potential to abstractly generalize the design space for arbitrary geometric inputs, even those beyond the scope of the training data.
Evaluation of the flow field of a marine vehicle reveals the underlying performance, however, the exact relationship between design features and their impact on the flow field is not well established. The goal of this work is first, to investigate the flow surrounding a self-propelled marine vehicle to identify the significance of various design decisions, and second, to develop a functional relationship between an arbitrary vehicle design and its flow field, thereby accelerating the design analysis process. Near-field wake profiles are computed through simulation, showing good agreement to experimental data. Machine learning is employed to discover the relationship between vehicle geometries and their associated flow fields with two distinct approaches. The first approach directly maps explicitly-specified geometric design parameters to their corresponding flow fields. The second approach considers the vehicle geometries themselves to implicitly-learn the underlying relationships. Once trained, both approaches generate a realistic flow field corresponding to a user-provided vehicle geometry, accelerating the design analysis from a multi-day process to one that takes a fraction of a second. The implicit-parameter approach successfully learns from the underlying geometric features, showing comparable performance to the explicit-parameter approach. With a larger and more-diverse training database, this network has the potential to abstractly learn the design space relationships for arbitrary marine vehicle geometries, even those beyond the scope of the training database.
To Hannah, my parents, and my brother.
Acknowledgments

I would like to thank my advisor Dr. Eric Paterson for his guidance and wisdom in academics and life. Thank you to my advisory committee members Dr. William Devenport, Dr. Christopher Roy, and Dr. Jonathan Pitt for providing valuable feedback to my work. Dr. Heng Xiao is acknowledged for providing inspiration to the machine learning formulation. I am grateful to the Rolls–Royce Commonwealth Graduate Fellowship at Virginia Tech for supporting part of my research. Finally, I would like to thank Newport News Shipbuilding – Huntington Ingalls Industries for their support in my research.
Contents

List of Figures xii

List of Tables xix

1 Introduction 1

2 Review of Literature 5

2.1 Low–Speed, Turbulent Wake Physics 5

2.1.1 Introduction 5

2.1.2 Wake Experiments 6

2.1.3 State–of–the–Art Simulation of Near and Far Fields 9

2.1.4 Future Prognosis 13

2.1.5 Conclusions 13

2.2 Vortex Breakdown 14

2.2.1 Introduction 14

2.2.2 Experimental Advancement 16

2.2.3 Theoretical and Computational Advancement 18

2.2.4 State–of–the–Art Tools and Future Advancement 20

2.2.5 Conclusions 21

2.3 Set–Based Design 21

2.3.1 Introduction 22

2.3.2 Origin and History 23
3 Validation and Scaling of the Stratified Near Wake of a Self-Propelled Body

3.1 Introduction .................................................. 56
3.2 Approach ...................................................... 58
  3.2.1 Governing Equations ................................. 58
  3.2.2 Kinetic and Potential Energies .................. 59
  3.2.3 Actuator-Line Model ................................. 59
  3.2.4 Iowa Body ............................................... 60
  3.2.5 Mesh ................................................... 62
  3.2.6 Flow Field Analysis ................................. 63
  3.2.7 Flow Coefficients and Case Studies ............ 63
  3.2.8 Characterizing Vortex Path ....................... 64
  3.2.9 Coupling Among Vortices ......................... 65
3.3 Comparison to Experiment ................................. 66
  3.3.1 Pressure Coefficient ................................. 66
  3.3.2 Vehicle Boundary Layer ............................. 66
  3.3.3 Phase-Averaged Contours ......................... 67
  3.3.4 Circumferentially-Averaged Wake Profiles .... 69
3.4 Results ...................................................... 71
  3.4.1 Buckingham II Dimensional Analysis ........... 71
  3.4.2 Reynolds Number Effects ........................... 72
  3.4.3 Characterizing Vortex Paths ...................... 73
  3.4.4 Coupling Among Vortices ......................... 74
3.4.5 Evolution of Kinetic and Potential Energies ............................................. 75
3.4.6 Evolution and Structure of the Mixed Patch ............................................. 76
3.4.7 Internal Froude Number Independence ..................................................... 78
3.4.8 Analytical Far–Field Profiles ................................................................. 79
3.4.9 Mixed–Patch Quasi–Analytical Profile ...................................................... 81
3.5 Conclusions .................................................................................................. 81
Bibliography ....................................................................................................... 83
Appendix A Analytical Profile for Mixed–Patch Temperature Deviation ............. 86

4 Influence of Propulsion Type on the Stratified Near Wake of a Self–Propelled Body 88
4.1 Introduction .................................................................................................. 89
4.2 Approach ..................................................................................................... 90
  4.2.1 Governing Equations ............................................................................. 90
  4.2.2 Kinetic and Potential Energy ................................................................. 92
  4.2.3 Actuator–Line Model ............................................................................ 93
  4.2.4 Iowa Body ............................................................................................ 93
  4.2.5 Iowa Body Propeller ........................................................................... 95
  4.2.6 Computational Mesh ............................................................................ 95
  4.2.7 Numerical Methods ............................................................................. 97
  4.2.8 Initial and Boundary Conditions ........................................................... 97
  4.2.9 Flow Field Analysis ............................................................................. 98
  4.2.10 Flow Coefficients and Case Studies ..................................................... 98
4.3 Results ........................................................................................................ 99
  4.3.1 Near–Wake Transition ......................................................................... 99
  4.3.2 Velocity Profiles .................................................................................... 101
  4.3.3 Velocity and Temperature Fields in the Mixed–Patch ......................... 102
  4.3.4 Potential and Kinetic Energy Evolution ............................................... 108
4.4 Conclusions ................................................................................................ 110
5 Influence of Hull–Form on the Stratified Near Wake of a Self–Propelled Body

5.1 Introduction

5.2 Approach

5.2.1 Governing Equations

5.2.2 Numerical Methods

5.2.3 Hull Geometry

5.2.4 Actuator Line Model

5.2.5 Computational Mesh

5.2.6 Initial and Boundary Conditions

5.2.7 Flow Conditions

5.2.8 Energy in the Flow

5.2.9 Vorticity Transport Equation

5.3 Results

5.3.1 Flow Visualization

5.3.2 Instantaneous and Time–Averaged Fields

5.3.3 Cross–Plane Profiles

5.3.4 Kinetic and Potential Energy

5.3.5 Vorticity–Transport Equation Terms

5.4 Conclusions

Bibliography

6 Deep Learning of the Near Wake with Explicitly–Specified Hull–Form Parameters

6.1 Introduction

6.2 Background

6.2.1 Artificial Neural Networks

6.2.2 Convolutional Neural Networks
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.3</td>
<td>Approach</td>
<td>143</td>
</tr>
<tr>
<td>6.3.1</td>
<td>Network Structure</td>
<td>143</td>
</tr>
<tr>
<td>6.3.2</td>
<td>Physics-Based Constraint: Divergence Free Flow</td>
<td>145</td>
</tr>
<tr>
<td>6.4</td>
<td>Results</td>
<td>146</td>
</tr>
<tr>
<td>6.4.1</td>
<td>Network Performance</td>
<td>146</td>
</tr>
<tr>
<td>6.5</td>
<td>Conclusions</td>
<td>150</td>
</tr>
<tr>
<td></td>
<td>Bibliography</td>
<td>151</td>
</tr>
<tr>
<td>7</td>
<td>Parameterized Hull Form: Influence of Cross-Section, Sail Appendage, and Slenderness</td>
<td>153</td>
</tr>
<tr>
<td>7.1</td>
<td>Introduction</td>
<td>154</td>
</tr>
<tr>
<td>7.2</td>
<td>Approach</td>
<td>156</td>
</tr>
<tr>
<td>7.2.1</td>
<td>Governing Equations</td>
<td>156</td>
</tr>
<tr>
<td>7.2.2</td>
<td>Numerical Methods</td>
<td>157</td>
</tr>
<tr>
<td>7.2.3</td>
<td>Actuator-Line Model</td>
<td>158</td>
</tr>
<tr>
<td>7.2.4</td>
<td>Parameterized Generic Hull Geometry</td>
<td>159</td>
</tr>
<tr>
<td>7.2.5</td>
<td>INSEAN E1619 Generic Propeller</td>
<td>161</td>
</tr>
<tr>
<td>7.2.6</td>
<td>CFD Database</td>
<td>164</td>
</tr>
<tr>
<td>7.2.7</td>
<td>Self-Propelled Body Computational Meshes</td>
<td>166</td>
</tr>
<tr>
<td>7.2.8</td>
<td>Kinetic and Potential Energy</td>
<td>166</td>
</tr>
<tr>
<td>7.3</td>
<td>Results</td>
<td>168</td>
</tr>
<tr>
<td>7.3.1</td>
<td>Open Water Propeller</td>
<td>168</td>
</tr>
<tr>
<td>7.3.2</td>
<td>Case Study: Body of Revolution with Sail Appendage</td>
<td>170</td>
</tr>
<tr>
<td>7.3.3</td>
<td>Effect of Cross-Sectional Aspect Ratio and Sail Appendage</td>
<td>176</td>
</tr>
<tr>
<td>7.3.4</td>
<td>Effect of Slenderness</td>
<td>181</td>
</tr>
<tr>
<td>7.4</td>
<td>Conclusions</td>
<td>184</td>
</tr>
<tr>
<td></td>
<td>Bibliography</td>
<td>190</td>
</tr>
<tr>
<td></td>
<td>Appendix B</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Contour Maps of Additional Flow-Field Variables</td>
<td>195</td>
</tr>
</tbody>
</table>
8 Deep Learning of the Near Wake with Implicitly–Learned Hull–Form Parameters

8.1 Introduction ......................................................... 203
8.2 Related Work ....................................................... 204
8.3 Approach .............................................................. 206
  8.3.1 Explicit–Parameter Network Structure ......................... 206
  8.3.2 Implicit–parameter Network Structure ......................... 208
  8.3.3 CFD Database ................................................. 214
8.4 Results ............................................................... 215
  8.4.1 Performance in Training ...................................... 216
  8.4.2 Performance in Inference ..................................... 219
  8.4.3 Performance in Prediction .................................... 220
  8.4.4 Performance for an Unconventional Geometry ............... 220
  8.4.5 Performance from the Perspective of Integrated Kinetic Energy . 222
8.5 Conclusions .......................................................... 223
Bibliography ............................................................. 225
Appendix C Generated Training Profiles .......................... 231

9 Conclusion and Future Work .............................................. 236
List of Figures

1.1 Physics–based simulation using the near–field $3D + t$ wake profile to initialize the far–field $2D + t$ simulation. ................................................................. 2
1.2 Illustration of variations in design. ................................................................. 2
1.3 Improvements in analysis for rapid evaluation. ................................................. 3

2.1 Photographs of bubble (a) and spiral modes (b) of vortex breakdown from experiment of Leibovich [124]. ................................................................. 15
2.2 Photograph of delta–wing setup from Lambourne and Bryer [122]. ............... 16
2.3 Photographs of double–helix breakdown at two different Reynolds and swirl numbers from Sarpkaya [197]. ................................................................. 17
2.4 Classical point–based design spiral of a marine vehicle following the criteria of Evans [63]. ................................................................. 26
2.5 SBD process described by Bernstein [18]. ........................................................... 27
2.6 Effect of SBD on product development cycle with notional curves inspired by Bernstein [18]. ................................................................. 28

3.1 Non–dimensional body, instantaneous force on mesh slice at propeller plane $f_p/(R_prps^2)$. ................................................................. 60
3.2 Iowa Body geometry. ................................................................. 60
3.3 Mesh generated with cfMesh [2]. ................................................................. 62
3.4 Process of extracting downstream positional data of root– and tip–vortices. ................................................................. 65
3.5 Pressure coefficient over unpropelled Iowa Body compared to the experimental data of Hyun and Patel [20]. ................................................................. 66
3.6 Boundary layer profiles on the body compared to the experimental data of Hyun and Patel [20]. ................................................................. 67
<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.7</td>
<td>Experimental–model–scale non–dimensional axial velocity $U_x/U_0$ at various axial locations compared to the experimental measurements of Hyun and Patel [20].</td>
<td>68</td>
</tr>
<tr>
<td>3.8</td>
<td>Experimental–model–scale non–dimensional axial vorticity $-\omega_z R_p/U_0$ at various axial locations compared to the experimental measurements of Hyun and Patel [20].</td>
<td>68</td>
</tr>
<tr>
<td>3.9</td>
<td>Experimental–model–scale non–dimensional turbulent kinetic energy $10^3 k/U_0^2$ at various axial locations compared to the experimental measurements of Hyun and Patel [20].</td>
<td>69</td>
</tr>
<tr>
<td>3.10</td>
<td>Circumferentially–averaged axial and swirl velocity profiles at $x/L = 1.01, 1.10, 1.30$.</td>
<td>70</td>
</tr>
<tr>
<td>3.11</td>
<td>Comparison of skin–friction coefficient between low and high Reynolds numbers.</td>
<td>73</td>
</tr>
<tr>
<td>3.12</td>
<td>$Q$–criterion flow visualization with $(L/U_0)^2 Q = 16.9$ extending to $x/L = 1.3$.</td>
<td>73</td>
</tr>
<tr>
<td>3.13</td>
<td>Radial and azimuthal position of root and tip vortices for $Re_L = 310 \times 10^6$.</td>
<td>74</td>
</tr>
<tr>
<td>3.14</td>
<td>Axial development of turbulent–kinetic, mean–kinetic, and potential energies.</td>
<td>75</td>
</tr>
<tr>
<td>3.15</td>
<td>Axial development of relevant wake features.</td>
<td>76</td>
</tr>
<tr>
<td>3.16</td>
<td>Instantaneous and time–averaged temperature deviation $\Delta T/\Delta T_{R_P}$ cross–plane surfaces at $x/L = 1.01, 1.10, 1.30, 1.45$.</td>
<td>77</td>
</tr>
<tr>
<td>3.17</td>
<td>Instantaneous iso–surface of the temperature at the background centerline value for $0.98 \leq x/L \leq 1.45$.</td>
<td>77</td>
</tr>
<tr>
<td>3.18</td>
<td>Instantaneous, on–dimensional temperature deviation $\Delta T/\Delta T_{R_P}$ at $x/L = 1.45$ compared to a theoretical, perfectly–mixed profile.</td>
<td>78</td>
</tr>
<tr>
<td>3.19</td>
<td>Center–plane depth–wise deviation to temperature $\Delta T/\Delta T_{R_P}$ for various stratification intensities at $x/L = 1.5$.</td>
<td>79</td>
</tr>
<tr>
<td>3.20</td>
<td>Instantaneous contour surfaces for $Re_L = 310 \times 10^6$ at $x/L = 1.45$.</td>
<td>80</td>
</tr>
<tr>
<td>3.21</td>
<td>Comparison between analytical and simulated profiles for $Re_L = 310 \times 10^6$ at $x/L = 1.45$.</td>
<td>80</td>
</tr>
<tr>
<td>3.22</td>
<td>Quasi–analytical model of the mixed–patch temperature–deviation field compared with simulated results for various azimuthal directions.</td>
<td>81</td>
</tr>
<tr>
<td>A.1</td>
<td>Variations in Fermat Spiral parameters.</td>
<td>87</td>
</tr>
<tr>
<td>4.1</td>
<td>Non–dimensional body force on mesh slice at propeller plane $</td>
<td></td>
</tr>
<tr>
<td>4.2</td>
<td>Standard Iowa Body profile.</td>
<td>94</td>
</tr>
<tr>
<td>Section</td>
<td>Description</td>
<td></td>
</tr>
<tr>
<td>---------</td>
<td>-------------</td>
<td></td>
</tr>
<tr>
<td>4.3</td>
<td>Computational meshes generated for each configuration.</td>
<td></td>
</tr>
<tr>
<td>4.4</td>
<td>Flow visualization for $x/L \leq 1.5$ using $Q$-criterion visualization non-dimensionalized as $(L/U_0)^2Q = 16.9$ colored by $U_x/U_0 - 1$ with vertical cutting plane through mesh.</td>
<td></td>
</tr>
<tr>
<td>4.5</td>
<td>Instantaneous (top) and time-averaged (bottom) velocity defect $U_x/U_0 - 1$ for the single propeller case.</td>
<td></td>
</tr>
<tr>
<td>4.6</td>
<td>Instantaneous (top) and time-averaged (bottom) velocity defect $U_x/U_0 - 1$ for the CRP case.</td>
<td></td>
</tr>
<tr>
<td>4.7</td>
<td>Circumferentially-averaged velocity defect profiles for each configuration at various distances downstream.</td>
<td></td>
</tr>
<tr>
<td>4.8</td>
<td>Circumferentially-averaged swirl velocity profiles for each configuration at various distances downstream.</td>
<td></td>
</tr>
<tr>
<td>4.9</td>
<td>Stationary velocity defect profiles at $x/L = 1.5$.</td>
<td></td>
</tr>
<tr>
<td>4.10</td>
<td>Stationary swirl velocity profiles at $x/L = 1.5$.</td>
<td></td>
</tr>
<tr>
<td>4.11</td>
<td>Single propeller $\Delta T/\Delta T_{R_p}$ at $x/L = 1.5$.</td>
<td></td>
</tr>
<tr>
<td>4.12</td>
<td>CRP $\Delta T/\Delta T_{R_p}$ at $x/L = 1.5$.</td>
<td></td>
</tr>
<tr>
<td>4.13</td>
<td>Jet configuration $\Delta T/\Delta T_{R_p}$ at $x/L = 1.5$.</td>
<td></td>
</tr>
<tr>
<td>4.14</td>
<td>Idealized mixed-patch profile of $\Delta T/\Delta T_{R_p}$ for a wake region of constant $T$.</td>
<td></td>
</tr>
<tr>
<td>4.15</td>
<td>Integrated energy evolution downstream of the vehicle for each configuration, with $x$ measured from the bow of the hull and $x'$ measured from the stern.</td>
<td></td>
</tr>
<tr>
<td>5.1</td>
<td>Standard axisymmetric Iowa Body profile compared to SUBOFF profile.</td>
<td></td>
</tr>
<tr>
<td>5.2</td>
<td>Three-dimensional and mid-body, cross-sectional views of Iowa Body hull-form variations.</td>
<td></td>
</tr>
<tr>
<td>5.3</td>
<td>Vertical cutting plane through mesh for $D_y/D_z = 1/1$.</td>
<td></td>
</tr>
<tr>
<td>5.4</td>
<td>Iso-surface of $(L/U_0)^2Q = 16.9$ and vertical cutting plane colored by $U_x/U_0 - 1$ for each hull form with $x/L &lt; 1.5$.</td>
<td></td>
</tr>
<tr>
<td>5.5</td>
<td>Comparison of instantaneous to time-averaged $U_x/U_0 - 1$ field for $D_y/D_z = 1/1$ at successive downstream locations.</td>
<td></td>
</tr>
<tr>
<td>5.6</td>
<td>Comparison of instantaneous to time-averaged $U_x/U_0 - 1$ field for $D_y/D_z = 1/2$ at successive downstream locations.</td>
<td></td>
</tr>
<tr>
<td>5.7</td>
<td>Comparison of $U_x/U_0 - 1$ at $x/L = 1.5$ for each towed hull form.</td>
<td></td>
</tr>
</tbody>
</table>
7.5 Cutting planes through coarse mesh of $5.3 \times 10^6$ cells.

7.6 Non-dimensional body force $\|f_p\|/(\rho_0 R_p \text{ rps}^2)$ on propeller plane for each mesh resolution.

7.7 Side view of the 27.4 million cell mesh extending to $x/L = 1.5$ for the $D_y/D_z = 1$, $b/c = 0.5$, and $L/D = 10$ geometry. Zoomed in views show cells focused in the sail, boundary layer, and propulsor regions.

7.8 Cross-sectional view at multiple axial stations of the 27.4 million cell mesh for the $D_y/D_z = 1$, $b/c = 0.5$, and $L/D = 10$ geometry.

7.9 INSEAN E1619 open-water propulsion coefficients.

7.10 Axial velocity $U_x/U_0$ comparison to LDV and LES of Liefvendahl et al. [35] at $x/R_p = 0.17$, clipped such that $\sqrt{y^2 + z^2} \leq R_p$.

7.11 Axial velocity $U_x/U_0$ comparison to LDV and LES of Liefvendahl et al. [35] at $x/R_p = 0.58$, clipped such that $\sqrt{y^2 + z^2} \leq R_p$.

7.12 Axial velocity $U_x/U_0$ comparison to LDV and LES of Liefvendahl et al. [35] at $x/R_p = 1.0$, clipped such that $\sqrt{y^2 + z^2} \leq R_p$.

7.13 Time-averaged velocity defect profiles in radial direction at three axial locations.

7.14 $Q$-criterion flow visualization of vortex structure with $(L/U_0)^2 Q = 1.5$ extending to $x/L = 1.5$ for the BOR geometry with a sail appendage.

7.15 Contours of instantaneous axial velocity defect $U_x/U_0 - 1$ at $x/L = 0.2$, 0.4, 0.6, 0.8, 1.0, 1.3, and 1.5 for the BOR geometry with a sail appendage.

7.16 Axial velocity defect $U_x/U_0 - 1$ for three mesh resolutions of the BOR geometry with a sail appendage at three $z$ locations: tip of the sail, half-span of the sail, and hull centerline.

7.17 Axial velocity defect and temperature perturbation for three mesh resolutions of the BOR geometry with a sail appendage at $x/L = 1.5$.

7.18 Comparison of instantaneous and time-averaged velocity defect $U_x/U_0 - 1$ at axial stations downstream of the body.

7.19 Vertical profiles of time-averaged velocity defect and temperature perturbation at several axial stations behind the BOR geometry with a sail appendage.

7.20 Side view of axial velocity defect $U_x/U_0 - 1$ for several hull-form and sail permutations with $L/D = 10$.

7.21 Maximum cross-sectional extent of various hull forms.

7.22 IDP axial velocity defect $U_x/U_0 - 1$ with variations in hull-form and sail aspect ratio for $L/D = 10$. 
7.23 IDP swirl velocity $U_\theta/U_0$ with variations in hull–form and sail aspect ratio for $L/D = 10$. .............................................. 182

7.24 IDP temperature perturbation $\Delta T/\Delta T_{R_p}$ with variations in hull–form and sail aspect ratio for $L/D = 10$. .............................................. 183

7.25 Side view of axial velocity defect $U_x/U_0 - 1$ for several hull–form and sail permutations and $L/D = 7$. .............................................. 185

7.26 IDP axial velocity defect $U_x/U_0 - 1$ with variations in hull–form and sail aspect ratio for $L/D = 7$. .............................................. 186

7.27 IDP swirl velocity $U_\theta/U_0$ with variations in hull–form and sail aspect ratio for $L/D = 7$. .............................................. 187

7.28 IDP temperature perturbation $\Delta T/\Delta T_{R_p}$ with variations in hull–form and sail aspect ratio for $L/D = 7$. .............................................. 188

B.1 IDP axial vorticity $\omega_x R_p/U_0$ with variations in hull–form and sail aspect ratio for $L/D = 10$. .............................................. 196

B.2 IDP turbulent kinetic energy $10^3(k - k_0)/U_0^2$ with variations in hull–form and sail aspect ratio for $L/D = 10$. .............................................. 197

B.3 IDP turbulence dissipation rate $10^3(\epsilon - \epsilon_0)/(U_0^3/R_p)$ with variations in hull–form and sail aspect ratio for $L/D = 10$. .............................................. 198

B.4 IDP axial vorticity $\omega_x R_p/U_0$ with variations in hull–form and sail aspect ratio for $L/D = 7$. .............................................. 199

B.5 IDP turbulent kinetic energy $10^3(k - k_0)/U_0^2$ with variations in hull–form and sail aspect ratio for $L/D = 7$. .............................................. 200

B.6 IDP turbulence dissipation rate $10^3(\epsilon - \epsilon_0)/(U_0^3/R_p)$ with variations in hull–form and sail aspect ratio for $L/D = 7$. .............................................. 201

8.1 Typical marine vehicle geometry. .............................................. 204

8.2 Diagram of the CNN with explicitly–specified geometric parameters. .............................................. 207

8.3 3D grid for extracting geometric volume fractions. .............................................. 209

8.4 Example of computation of geometric volume fractions for the $D_y/D_z = 1.0$, $b/c = 0.5$, and $L/D = 10$ geometry. .............................................. 209

8.5 Example of computation of geometric volume fractions for the $D_y/D_z = 2.25$, $b/c = 0.55$, and $L/D = 11$ geometry. .............................................. 210
8.6 Overall cGAN layout: the discriminator learns from training and generated wake profiles, while the generator tries to fool the discriminator. Geometries are input as volume fraction tensors, so the underlying geometric parameters must be learned implicitly.

8.7 Diagram of the U–Net generator for the cGAN with implicitly–learned geometric parameters.

8.8 Diagram of the discriminator CNN for the cGAN.

8.9 Computation of geometric volume fractions for an unconventional geometry with $b/c = 0.5$, and $L/D = 10$, $D_y/D_z = 1.0$ on port side, and $D_y/D_z = 2.0$ on starboard side.

8.10 Three evaluation cases: inference, prediction, and an unconventional geometry outside of the scope of the training data.

8.11 Explicit–parameter CNN training and test error.

8.12 Pre–training of U–Net generator showing training and test error.

8.13 Adversarial training losses of the implicit–parameter cGAN generator and discriminator compared to test error.


8.15 Implicit–parameter cGAN performance in inference.

8.16 Explicit–parameter CNN performance in prediction.

8.17 Implicit–parameter cGAN performance in prediction.

8.18 Implicit–parameter cGAN performance for an unconventional geometry outside of the scope of the training data.

C.1 Explicit–parameter CNN generated training profiles and error for $L/D = 10$ geometries.

C.2 Explicit–parameter CNN generated training profiles and error for $L/D = 7$ geometries.

C.3 Implicit–parameter cGAN–generated training profiles and error for $L/D = 10$ geometries.

## List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Low-speed, turbulent wake experiments.</td>
<td>7</td>
</tr>
<tr>
<td>3.1</td>
<td>Iowa Body geometry.</td>
<td>61</td>
</tr>
<tr>
<td>3.2</td>
<td>Propeller blade radial geometry including chord, thickness, camber, pitch,</td>
<td>61</td>
</tr>
<tr>
<td></td>
<td>and twist.</td>
<td></td>
</tr>
<tr>
<td>3.3</td>
<td>Mesh design and quality features.</td>
<td>63</td>
</tr>
<tr>
<td>3.4</td>
<td>Grid study.</td>
<td>63</td>
</tr>
<tr>
<td>3.5</td>
<td>Flow coefficients.</td>
<td>64</td>
</tr>
<tr>
<td>3.6</td>
<td>Case studies.</td>
<td>65</td>
</tr>
<tr>
<td>3.7</td>
<td>Correlation between root and tip vortical paths</td>
<td>74</td>
</tr>
<tr>
<td>3.8</td>
<td>Power law with $E = a(x/L)^b$.</td>
<td>76</td>
</tr>
<tr>
<td>4.1</td>
<td>Coefficients in UNESCO equation of state for seawater.</td>
<td>92</td>
</tr>
<tr>
<td>4.2</td>
<td>Iowa Body geometry.</td>
<td>94</td>
</tr>
<tr>
<td>4.3</td>
<td>Mesh design and quality features.</td>
<td>96</td>
</tr>
<tr>
<td>4.4</td>
<td>Flow coefficients.</td>
<td>98</td>
</tr>
<tr>
<td>5.1</td>
<td>Mesh features.</td>
<td>119</td>
</tr>
<tr>
<td>6.1</td>
<td>Hyperparameters for network.</td>
<td>145</td>
</tr>
<tr>
<td>7.1</td>
<td>Marine–vehicle, turbulent wake experiments.</td>
<td>156</td>
</tr>
<tr>
<td>7.2</td>
<td>Geometric ratios for the generic hull form.</td>
<td>159</td>
</tr>
<tr>
<td>7.3</td>
<td>Shape factors assigned to the generic hull form.</td>
<td>160</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

The acquisition and life-cycle of a marine vehicle requires an extensive multidisciplinary effort to achieve desirable cost, safety, and performance. Many areas of specific requirements feed into the design process such as hydrostatics/hydrodynamics, weight/space/volume/arrangements, structures, propulsion, and even operational capabilities. One method of evaluating the hydrodynamics involves examining the imparted flow field of the self-propelled vehicle, which provides insight into the vehicle’s efficiency and performance. At the present time, however, the relationship between the marine vehicle propulsor and hull-form choices and the imparted flow field are not well understood.

For efficiency and tractability, the simulation of turbulent wake physics is typically divided into the near-field and far-field spatial regimes as illustrated in Figure 1.1. Simulation of the near field requires resolving the 3-dimensional geometry of the self-propelled vehicle in operation. Computational Fluid Dynamics (CFD) high-performance computing capabilities limit the downstream extent of the unsteady, geometry-resolved “$3D+t$” simulation to $O(L)$ where $L$ is the vehicle body length, so it is difficult to investigate extensive far-field physics in this type simulation. Simplifications make spatially-large far-field simulations feasible. One approach is to solve the parabolized Navier-Stokes equations over a 2D computational domain in time, initialized by a developed wake profile from the $3D+t$ simulation. Under the assumption that axial gradients are small, these “$2D+t$” simulations yield the evolution of the wake over large spatial and temporal scales, $O(L^3)$, at an efficient computational cost. The far-field evolution of the wake in the surrounding environment can then be studied.

The difficulty of engineering design is well-established. Being an open-ended problem, there are many possible paths to reach the final product. The conceptual design phase is particularly important because uncertainty is largest, information is scarce, and decisions have the greatest impact. Numerous concept variations are considered in an attempt to identify the best designs moving forward, as shown in Figure 1.2. Sometimes the conceptual design phase requires numerous $O(10^4)$ concept iterations, especially within the paradigms of Multidisciplinary Design Optimization (MDO) or set-based design (SBD). Consequently, the
end–to–end $3D + t$ to $2D + t$ simulation is prohibitively expensive for these methodologies. Near–field simulations takes several days on $O(10^3)$ cores. Far–field simulations are faster, but may take several hours on $O(10^2)$ cores. Understanding these limitations, the objective of this research is clear: develop a fast algorithm that will generate a realistic flow–field wake profile for the evaluation of important design metrics while bypassing the cost of online CFD simulations.

To achieve this objective, data–driven machine–learning algorithms are trained to generate developed flow–field wake profiles for arbitrary, simplified vehicle concepts. This process essentially eliminates the use of CFD in preliminary design except for the offline generation of training data. Deep–learning Convolutional Neural Networks (CNNs) are trained from a database of flow–field wake profiles extracted from $3D + t$ simulations. The first algorithm explicitly takes geometric parameter inputs to generate the appropriate flow–field wake profile for the vehicle concept. The second algorithm implicitly–learns relevant geometric parameters from the training database. For the latter case, the entire vehicle geometry, itself, is input into the deep–learning algorithm as a unique tensor of discrete volume fractions. This network is implemented as a conditional Generative Adversarial Network (cGAN) with a U-Net generator and a CNN discriminator that compete with one another in training. The generator seeks to fool the discriminator by generating realistic flow–field wake profiles,
(a) Physics–based simulation and MDO. (b) Acceleration of analysis.

Figure 1.3: Improvements in analysis for rapid evaluation.

while the discriminator tries to discern between those fake profiles and the real profiles of the training data.

Both of these algorithms accomplish their goal in accelerating the design analysis as portrayed in Figure 1.3. Instead of running expensive $3D + t$ simulations for every concept of interest, the trained CNN can instantly generate a flow–field wake profile, which can then be evaluated through some performance metric. There are many metrics which feed into the design process, for example propulsive efficiency or hydrodynamic resistance, and it is the task of the designer to evaluate the appropriate metrics to meet their requirements.

The overall contributions of this work are listed.

1. The literature is thoroughly reviewed for three topics: the experimentation and simulation of low–speed, turbulent wakes, the vortex breakdown phenomenon, and the set–based design paradigm.

2. The novel implementation of actuator–line–modeled propellers in the CFD simulation of a self–propelled vehicle is validated against experimental data, and scaling effects are shown.

3. The process of automatic mesh generation to near–field CFD simulation was implemented for over 50 distinct complex geometries.

4. The influences of propulsion scheme and hull form on the stratified near–field evolution of energies and flow properties are identified.

5. Two data–driven machine–learning models are implemented to provide the functional mapping between a vehicle geometry and its flow–field wake profile, one taking explicit geometric–parameter inputs and the other implicitly–learning the geometric parameters from the input geometries, themselves.

6. A conditional Generative Adversarial Network with a U-Net generator is implemented to implicitly–learn geometric parameters, showing promise for predicting flow fields even outside of the scope of the training data.
Chapter 1. Introduction

This dissertation is organized as follows. First, in Chapter 2, the literature is reviewed to identify the state–of–the–art in experimentation and simulation of marine vehicle turbulent wakes. Literature is surveyed regarding the vortex breakdown phenomenon because of its importance in the near–field wake evolution. The set–based design process is also surveyed to understand how these deep learning approaches can accelerate modern design analysis. Chapter 3 is dedicated to the validation of experimental–model–scale near–field simulations and the effects of scaling to larger Reynolds and Froude numbers. The effect of propulsion type is investigated in Chapter 4, where the high–Reynolds–Number near–field is analyzed for a swirl–free jet, standard propeller, and contra–rotating propellers. Chapter 5 considers the influence of variations in hull–form cross–sectional aspect ratio. In Chapter 6, the explicit deep–learning CNN is formulated and evaluated for the hull–form variation data set. A new, generic hull form is described in Chapter 7, which allows for the addition of a sail appendage and can be easily modified to provide numerous variations in geometry. Near–field simulations of this geometry are conducted with variations in cross–sectional aspect ratio, sail appendage size, and length–to–diameter ratios. The flow–field wake profiles taken from these simulations are used as ground–truth data for the CNN that implicitly–learns the geometric parameters of vehicle concepts. This network is evaluated in Chapter 8.
Chapter 2

Review of Literature

The literature is reviewed for several topics important to the contemporary design of marine vehicles. First, the history of state–of–the–art experimentation and simulation into low–speed, turbulent wake physics is presented. Following is a review of the fluid–flow vortex–breakdown phenomenon because of its significance in the transition from near–field to far–field wake regimes. Finally, the contemporary set–based design process is discussed to illustrate how machine–learning–aided design analysis is well–suited to the modern conceptual design process. For each review, summary points and a prognosis for the future are given.

2.1 Low–Speed, Turbulent Wake Physics

This review is divided into two parts. The first part covers the history and advancement in low–speed, turbulent wake experiments. The second part covers advances in the state–of–the–art of simulation techniques, considering near– and far–field domains. Conclusions are drawn and key points are summarized.

2.1.1 Introduction

A surge of interest into low–speed, turbulent wake physics began in the 1960s. Experimentalists would study the evolution of wake properties and identify self–similarity in drag and net–zero–momentum wakes in model–scale experiments. The influence of a stratified environment on the wake evolution was of key interest because of the presence of stratification in the ocean. In the 1970s, advances in fluid computation allowed for the numerical simulation of simplified wakes initialized with analytical momentum distributions. To contrast, the modern state–of–the–art in computation allows for high–fidelity near–field simulations over
geometrically–resolved, self–propelled vehicles. While experiments provide valuable data on fundamental physics, Computational Fluid Dynamics (CFD) can provide insight into wake phenomena in situations that are difficult to experiment on.

2.1.2 Wake Experiments

A large body of work exists in the experimental investigation of wake physics. Table 2.1 lists important published experiments for low–speed, turbulent wakes, both towed and self–propelled. These experiments include stratified and non–stratified experiments. The determination of applicable experiments is in some ways subjective. Some experiments use notional marine–vehicle geometries, while others measure the wake behind simplified geometries such as disks or spheres, still providing valuable insight into wake physics. Due to practical challenges, there is a lack of ship–scale experimental data to provide insight into scale effects. Columns within the table list the author(), year, geometry, type of data taken, type of wake momentum, presence of stratification, and instrumentation used. Within the “Type of Data” column, properties refers to turbulence properties. Within the “Momentum” column, “drag” refers to towed experiments without self–propulsion. NNM, NZM, and NPM refer to propelled wakes with a net–negative, net–zero, and net–positive integrated momentum, respectively. The “Instrumentation” column includes the acronyms: hot wire anemometry (HWA), laser Doppler velocimetry (LDV), and particle image velocimetry (PIV).


Stockhausen et al. [230] measured salinity profiles behind a 27 × 4.5 inch BOR with a frontal plate and dual propellers. Several researchers measured flow properties behind simple towed geometries [27, 29, 42]. In 1973, Lin and Pao [133] studied the stratified wake collapse of a towed slender body and later a self–propelled slender body using shadowgraph and dye visualizations [169]. A large body of experimental work would follow. One year later, Lin, Pao, and Veenhuizen would measure turbulence properties of the wake using HWA and conductivity probes [134, 169]. Lin and Pao [132] provides a review of stratified wake experiments. Following their work, Schetz and other researchers would investigate realistic slender body geometries in both towed and self–propelled configurations, evaluating the
### Table 2.1: Low-speed, turbulent wake experiments.

<table>
<thead>
<tr>
<th>Author</th>
<th>Year</th>
<th>Geometry</th>
<th>Type of Data</th>
<th>Momentum</th>
<th>Stratification</th>
<th>Instrumentation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hall &amp; Bislop [85]</td>
<td>1938</td>
<td>2:1 Cylinder</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Cooper &amp; Lutzky [45]</td>
<td>1955</td>
<td>Thin disks</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Ilizarova &amp; Pochinka [104]</td>
<td>1962</td>
<td>6:0.71 BOR</td>
<td>Mean flow</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Schooley &amp; Stewart [203]</td>
<td>1963</td>
<td>2:2 cm axisymmetric motor attached to 1:8 cm propeller</td>
<td>Wake dimensions</td>
<td>NZM</td>
<td>Stratiﬁed</td>
<td>Dye vis.</td>
</tr>
<tr>
<td>Rijoranovic [192]</td>
<td>1963</td>
<td>DWCJ</td>
<td>Mean flow &amp; properties</td>
<td>NZM</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Carmody [30]</td>
<td>1964</td>
<td>Disk</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Wang [248]</td>
<td>1965</td>
<td>DWCJ</td>
<td>Mean flow &amp; properties</td>
<td>NZM</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Naalascher [160]</td>
<td>1965</td>
<td>DWCJ</td>
<td>Mean flow &amp; properties</td>
<td>NZM</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Stockhausen, Clark, &amp; Kennedy [230]</td>
<td>1966</td>
<td>27 in × 4.5 in BOR w/ frontal plate &amp; dual propellers</td>
<td>Salinity proﬁles</td>
<td>NZM</td>
<td>Stratiﬁed</td>
<td>Conductivity</td>
</tr>
<tr>
<td>Ginevskii, Pochinka, &amp; Ukhovskova [78]</td>
<td>1966</td>
<td>DWCJ</td>
<td>Mean flow &amp; properties</td>
<td>NZM</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Buchinskaya &amp; Pochina [27]</td>
<td>1968</td>
<td>6:1 Ellipsoid</td>
<td>Mean flow</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Chevray [42]</td>
<td>1968</td>
<td>6:1 Spheroid</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Gibson, Chen, &amp; Lin [37]</td>
<td>1968</td>
<td>Sphere</td>
<td>Properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Uchiti &amp; Freymuth [243]</td>
<td>1969</td>
<td>Sphere</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Bukreev, Kostomakha, &amp; Lytkin [29]</td>
<td>1972</td>
<td>8:1 slender body</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Holensone &amp; Schetz [95]</td>
<td>1975</td>
<td>Sphere</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Gran [80]</td>
<td>1975</td>
<td>Rankine ovoid w/ propeller</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Bukreev, Kostomakha, &amp; Lytkin [28]</td>
<td>1975</td>
<td>Slender body w/ jet or propeller</td>
<td>Mean flow &amp; properties</td>
<td>Drag &amp; NZM</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Chiang, Jakubowski, &amp; Schetz [43]</td>
<td>1975</td>
<td>Slender body w/ jet or propeller</td>
<td>Mean flow &amp; properties</td>
<td>Drag &amp; NZM</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Hokenson &amp; Schetz [95]</td>
<td>1975</td>
<td>Sphere</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Lin et al. [135, 136]</td>
<td>1974</td>
<td>Sphere Internal waves</td>
<td>Internal waves</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>Shadowgraph</td>
</tr>
<tr>
<td>Bonneton, Chomaz, and Hopfinger [20]</td>
<td>1993</td>
<td>Sphere</td>
<td>Internal waves</td>
<td>Drag</td>
<td>Stratiﬁed</td>
<td>Shadowgraph</td>
</tr>
<tr>
<td>Spedding, Browand, &amp; Fincham [226, 227, 228]</td>
<td>1996</td>
<td>Sphere</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>PIV</td>
</tr>
<tr>
<td>Faure &amp; Robert [64]</td>
<td>1996</td>
<td>Sphere w/ jet</td>
<td>Mean flow</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; LDV</td>
</tr>
<tr>
<td>Chemezas et al. [41, 255]</td>
<td>1997</td>
<td>6:1 prolate spheroid</td>
<td>Mean flow</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>LDV</td>
</tr>
<tr>
<td>Cheppon &amp; Babenko [165]</td>
<td>1998</td>
<td>Afoil with jet</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Gavrilov et al. [76]</td>
<td>2000</td>
<td>BOR w/ jet</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Bonnier, Eiff, and Bonneton [21]</td>
<td>2000</td>
<td>Sphere</td>
<td>Density proﬁle</td>
<td>Drag</td>
<td>Stratiﬁed</td>
<td>Conductivity</td>
</tr>
<tr>
<td>Johansson &amp; George [110, 111, 112, 113]</td>
<td>2002</td>
<td>Disk</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Meunier &amp; Spedding [154]</td>
<td>2004</td>
<td>Cylindrical, 6:1 prolate spheroid, sphere, hemisphere, &amp; disk</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Meunier &amp; Spedding [155]</td>
<td>2006</td>
<td>Cylinder &amp; prolate spheroid</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Voropayev et al. [246]</td>
<td>2007</td>
<td>75 cm submarine model</td>
<td>Surface plane</td>
<td>NZM &amp; NPM</td>
<td>Stratiﬁed</td>
<td>PIV &amp; conductivity</td>
</tr>
<tr>
<td>DeMoss &amp; Simpson [50, 51]</td>
<td>2007</td>
<td>NNEMO</td>
<td>Mean flow</td>
<td>NZM</td>
<td>Non-stratified</td>
<td>7-hole probe</td>
</tr>
<tr>
<td>Voropayev &amp; Fernando [247]</td>
<td>2010</td>
<td>10 cm submarine model</td>
<td>Surface plane</td>
<td>NZM &amp; NPM</td>
<td>Stratiﬁed</td>
<td>Dye vis. &amp; PIV</td>
</tr>
<tr>
<td>Anderson et al. [5]</td>
<td>2013</td>
<td>SUBOFF BOR</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>HW &amp; PIV</td>
</tr>
<tr>
<td>Manshadi et al. [145]</td>
<td>2015</td>
<td>SUBOFF BOR &amp; appended</td>
<td>Mean flow</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>5-hole probe</td>
</tr>
<tr>
<td>Furey et al. [73]</td>
<td>2016</td>
<td>DSTO Joubert appended</td>
<td>Mean flow</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>PIV</td>
</tr>
<tr>
<td>Burt et al. [92, 257, 258]</td>
<td>2016</td>
<td>Torpedo-like</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>PIV</td>
</tr>
<tr>
<td>Hayder et al. [93]</td>
<td>2018</td>
<td>Torpedo-like</td>
<td>Mean flow &amp; properties</td>
<td>Drag</td>
<td>Non-stratified</td>
<td>PIV</td>
</tr>
</tbody>
</table>
impact of jet–driven versus propeller–driven vehicles [43, 46, 199, 200, 236]. Schetz and Jakubowski [199] provide a tabulated review of low speed, axisymmetric turbulent wake experiments prior to 1975. Merritt [152] measured the spatial evolution of a stratified NZM wake created from an oscillating grid. Huang et al. [97, 98] would evaluate the impact of stern geometry on the near–field wake profiles for towed and self–propelled configurations. Swean and Schetz [237] measured temperature distributions behind a propeller–driven slender body with a sail to 4 body diameters downstream. Temperature in the wind tunnel experiment was linearly stratified through the injection of heated air in a manner that preserved uniform velocity in the freestream. A brief hiatus in marine–vehicle wake experiments would follow.

In 1989, the Defense Advanced Research Projects Agency (DARPA) SUBOFF project established a parametric axisymmetric geometry with stern appendages that would be used as a basis for CFD computations [84, 84, 137]. Huang et al. [96, 239] conducted experiments for the towed SUBOFF geometry. While these experiments provide the stern–boundary layer pressures, velocities, skin friction and Reynolds stresses, they do not provide wake–evolution data nor are they self–propelled.

In 1990, Hyun and Patel [100, 101, 102] conducted a wind tunnel experiment on the Iowa Body. To the author’s knowledge, this was the first and only experiment with phase–averaged measurements in the turbulent wake of a propeller–driven NZM vehicle. Contour maps of the wake profiles show the transition from the complicated region directly behind the propeller to half of a body length downstream where wake profiles have mixed and become axisymmetric. Cimbala and Park [44, 170] studied the wake behind a jet–propelled 2D airfoil. By evaluating wakes with net–negative, net–zero, and net–positive momentum, they showed the sensitivity of wake structure to budgets or deficits in momentum. Higuchi and Kubota [94] would also study this sensitivity in the experiment of a thin–walled tube with a jet. Spedding et al. [226] investigated self–similarity and the far–field wake evolution of a towed sphere in a stratified fluid. Faure and Robert [64] looked at the self–similarity of kinetic energy of a self–propelled body wake. Using the Iowa Body geometry, Sirviente and Patel [207, 208, 209, 210, 211] examined the influence of swirl on the evolution of a NZM wake. The Iowa Body was propelled using a jet with and without swirl. Gavrilov et al. [76] also studied a BOR with a jet. Bonnier et al. [21] found the vertical density profile behind a towed sphere in linear stratification.

In 2004, Meunier and Spedding [154] showed similarities in stratified wake profiles between various towed geometries: a disk, a hemisphere, a sphere, a cube, a 6:1 axially towed cylinder, and a 6:1 prolate spheroid. They found scaling exponents that they claimed could be applied to all stratified drag wakes to describe the mean flow, vortex geometry, and turbulence quantities. Meunier and Spedding [155] would later compare the self–propelled 6:1 cylinder and spheroid geometries finding that such similarity in scaling exponents did not exist for NZM wakes.

In 2007, Voropayev et al. [246] used PIV to measure surface disturbances from both a sub–surface self–propelled body and a turbulent jet in a two–layer stratified fluid. Using a sim-
2.1. Low-Speed, Turbulent Wake Physics

ilar experimental setup, Voropayev and Fernando [247] would later show how maneuvering self-propelled bodies in stratified fluids could impart large, long-lasting, coherent vortices. DeMoss and Simpson [50, 51] studied the surface flow and wake of the towed, non-BOR ellipsoidal geometry known as the Newport News Experimental Model One (NNEMO).

After 2010, additional measurements were taken for the towed SUBOFF geometry in both appended and appendageless configurations. Jiménez et al. [109] showed how stern appendages modified the near-field wake profiles within 9 body diameters downstream. Jiménez et al. [108] would also show the influence of Reynolds number for the appendageless configuration. Ashok, Buren, and Smits [6, 7, 8, 9] investigated the wake of the towed SUBOFF geometry in both yaw and pitch conditions. Although SUBOFF has functioned as a useful open-literature submarine, it is criticized for having a pure BOR hull and having relatively small appendages compared to the hull length. The Defense Science and Technology Organisation (DSTO) Joubert conventional submarine geometry was devised to be more realistic. Anderson et al. [5] took measurements of flow around the hull and immediately downstream of the towed DSTO Joubert in both barehull and appendage configurations. Fureby et al. [73] took PIV measurements in axial planes on and close to the DSTO Joubert body at a 10° yaw angle. Most recently, some researchers are studying flow effects around a torpedo-like geometry [92, 93, 257, 258].

2.1.3 State-of-the-Art Simulation of Near and Far Fields

Simulation of low-speed, turbulent wake physics is ordinarily divided into two spatial regimes: the near field and far field. The near-field simulation often involves resolving the 3-dimensional geometry of the self-propelled vehicle and its propulsor in time. Computational capabilities limit the downstream extent of the unsteady, geometry-resolved “3D + t” simulation, so it is difficult to investigate extensive far-field physics in this type simulation. Simplifications are made to make very large far-field simulations feasible.

Far Field

In 1974, Lewellen et al. [128] solved the steady, 2D Reynolds-Averaged Navier-Stokes (RANS) equations with a second-order turbulence closure applied to axisymmetric wakes with excess momentum and net-zero momentum. Results were compared to the experimental measurements of Naudascher [160] and Chevray [42]. Lewellen et al. [127] would later expand their approach to solve the steady, 3-dimensional Navier-Stokes equations to investigate a variety of turbulent wakes with excess momentum and NZM, with and without swirl, and in both stratified and non-stratified environments. Finson [69] applied a similar approach to study the similarity behavior of a NZM wake. For simplified, non-stratified, boundary-free shear flows it is possible to derive analytical power-law growth and decay coefficients [238]. Experimental and computational growth and decay rates are often compared to these power
One approach to solving spatially–large far–field simulations is to solve the parabolized Navier Stokes equations over a 2D computational mesh in time. A 2D wake profile is used to initialize these simulations. With the assumption that axial gradients are small, these “2D+t” simulations yield the evolution of the wake over large spatial and temporal scales at a relatively small computational cost. In 1980, Hassid [91] used this 2D+t approach to simulate the evolution of both drag and NZM wakes in a stratified fluid, comparing to the experiments of Lin and Pao [134, 169]. Hassid demonstrated the periodic wake collapse behavior of a stratified wake, which suppresses vertical growth and enhancing horizontal growth while also modifying flow–field decay rates. Chernykh et al. [39] would seek to improve the numerical modeling techniques of a towed wake in a stratified fluid. In 1999 Chernykh and Voropayeva [38] studied NZM wakes in stratified fluids using the 2D + t approach and comparing the performance of a selection of semi–empirical second–order turbulence models. They showed good comparison to the experiments of Lin and Pao [134, 169]. Jones and Paterson [115] used the 2D + t approach to study the influence of environmental forcing on the evolution of three notional boundary–free shear flows: jet, drag, and NZM, finding that Stokes–Ekman forcing significantly alters the far–field wake evolution in a stratified environment.

Other researchers have sought to leverage direct numerical simulation (DNS) to study the fundamental physics of stratified wakes at lower Reynolds numbers. In 2001, Gourlay et al. [79] first employed spatially 3–dimensional DNS in the simulation of non–stratified and stratified towed wakes, providing insight into the formation of “pancake eddies”. With a Reynolds number of $10^4$, they showed good agreement to previous experimental work such as that of Spedding et al. [226] to justify their conclusions. Johansson et al. [111, 114] would use this DNS data combined with previous experimental data to quantify the existence of similarity regimes for both low and high Reynolds numbers. Although not using DNS, Dommermuth et al. [55] used 3D + t Large Eddy Simulation (LES) to simulate a towed sphere in stratification for Reynolds numbers of $10^4$ and $10^5$, finding similar results to that of Gourlay et al. [79]. In 2010, Brucker and Sarkar [26] used DNS to study drag and NZM wakes in a stratified fluid for a Reynolds number of $5 \times 10^4$. The NZM wake decayed more quickly than the drag wake due to a larger shear production of turbulence. For the NZM case, they showed the existence of a non–equilibrium stage in the transition from the near to the quasi–two–dimensional (Q2D) far field, which was previously established for towed wakes. They also showed the importance of buoyancy in the NZM case because it decoupled regions of negative and positive momentum which, in turn, decay at different rates. de Stadler et al. [49] used DNS for a Reynolds number $10^4$ to study the effect of Prandtl number on a towed sphere in a stratified environment, finding that the Prandtl number has a significant effect on the perturbation of density. Using the same method, de Stadler and Sarkar [48] examined an accelerating, self–propelled, stratified wake, and compared it to a typical NZM wake. Although excess momentum in the NPM wake increased velocity defect, shear, kinetic energy, and wake width, both the NPM and NZM wakes shared qualitative similarities in their evolution. Chernykh et al. [40] and Voropaeva et al. [245] showed that
RANS with a second–order turbulence model provides a comparable solution to that of DNS when considering stratified drag wake decay rates. Through DNS in a range of Reynolds numbers, Redford et al. [189, 190] investigated the three stratified drag wake regimes: 3–dimensional near field, non–equilibrium wake collapse, and Q2D far field. They showed how buoyancy modifies turbulence during the wake collapse and analyzed the energy budgets of mean axial momentum, kinetic energy, and turbulent kinetic energy in each of the three regimes. Pal et al. [166, 167, 168] would compare the wake behind a towed sphere in stratified and non–stratified environments through DNS at a Reynolds number of $3.7 \times 10^3$, highlighting the influence of buoyancy throughout the wake evolution.

Near Field

Although DNS has proven to be an invaluable tool for investigating fundamental flow physics, it is too computationally expensive to apply to the higher Reynolds–number flows of most practical applications. In 2014, Slotnick et al. [212] predicted the use of CFD methods in simulating aerodynamic flows until the year 2030. The most significant limitation is the accurate simulation of turbulent flows, a difficulty that RANS methods will have difficulty overcoming due to large modeling uncertainties [256]. Wall–resolved LES provides much higher fidelity but at a much higher expense. So, with increasing computational power, researchers will likely transition from using RANS methods to LES methods. For the near future, hybrid RANS/LES methods are desirable because they allow for more accurate turbulence simulation at a lesser expense than pure LES. Bensow [17] surveyed CFD methods particular to the simulation of ship hydrodynamics. While potential flow methods have provided computationally efficient solutions to ship design, the present state–of–the–art for simulating self–propelled vehicles in operation requires the more advanced RANS, LES, and hybrid RANS/LES methods.

Of particular difficulty is simulating unsteady turbulence around a propeller. In 2009, Di Felice et al. [53] analyzed the wake behind the 7–bladed INSEAN E1619 propeller, comparing LES to experimentally–obtained data. Liefvendahl et al. [129] investigated the same propeller, also comparing LES to experimental data. The work of Felli et al. [66], while purely experimental, is mentioned because of its role in identifying the physical mechanisms behind the breakdown of a propeller wake. Peng et al. [179] compared various turbulence models in the RANS calculations of an open–water propeller. Chase and Carrica [34, 35] analyzed the open–water performance of the INSEAN E1619 propeller comparing RANS, Delayed Eddy Simulation (DES), and Delayed Detached Eddy Simulation (DDES) methods to each other and experimental data. They found the RANS method to be overly dissipative. In 2015, Balaras et al. [12] applied the immersed boundary (IB) method with LES to simulate the complicated interactions between propeller vortex structures. More recently, Kumar and Mahesh [121] used LES to identify a mutual–induction instability mechanism where tip vortices interact with vortex structures generated off of the blade trailing edge. Using RANS in 2018, Wang et al. [250] showed how increasing skew ultimately delayed the wake breakdown.
by modifying the interactions between vortex structures. Posa et al. [181] demonstrated the suitability of LES for submarine propellers by simulating the INSEAN E1619 propeller using LES and showing good agreement to the PIV measurements of Felli and Falchi [65].

Computational difficulty is significant for simulating self–propelled vehicles in operation due to inherent complexities such as: high Reynolds numbers, complicated geometric features, moving machinery, and oceanic physical phenomena. Several researchers have evaluated the costs and benefits of numerical methods on practical near–field calculations. In 2009, Karls-son and Fureby [116] compared LES, DES, RANS to experimental data for the towed 6:1 prolate spheroid reported in Chesnakas et al. [41] at a 20° incidence, demonstrating LES and DES to have the best performance. Alin et al. [2] compared same three methods for the SUBOFF hull both with and without appendages in towed and self–propelled configurations. They found that all methods compared well to experiment, however LES best–modeled the complicated flow in the self–propelled, appended configuration. Anderson et al. [5] compared RANS and LES methods to support their Joubert experiment in 2012, finding that LES performs better in the presence of appendages. Concurrent with their open–water simulations, Chase and Carrica [34, 35] simulated the appended DARPA SUBOFF geometry in self propulsion with the INSEAN E1619 propeller using RANS, DES, and DDES. Analyzing the structure of the near–field wake immediately behind the propeller they found DES and DDES to work perform best. They also studied the SUBOFF geometry in maneuvers [36]. Kim and Rhee [119] compared Improved Delayed Detached Eddy Simulation (IDDES) to Very Large Eddy Simulation (VLES) in the simulation of the self–propelled, appended SUBOFF hull, finding that both methods needed to be improved in the near future until wall–resolved LES could become feasible for ship–scale applications. Alongside their experimental work in 2016, Fureby et al. [73] simulated the model–scale, towed Joubert submarine in straight–ahead and yaw configurations using both RANS and LES with a 340 million cell mesh. LES was found to better–predict vortex structures. Liefvendahl and Fureby [130] examined the grid requirements for wall–modeled and wall–resolved LES at both model and ship scales, illustrating the computational challenges in simulating flows around realistic geometries. While RANS can provide reasonable flow fields, LES methods better–approximate the origin and evolution of vortex structures.

In the present state–of–the–art of simulating operating self–propelled undersea vehicles, LES and hybrid LES methods provide the highest fidelity. In 2016, Esmaeilpour et al. [62] simulated the stratified flow around the self–propelled Joubert BB2 submarine and a surface ship using DDES. They also simulated a stratified lid–driven cavity and towed sphere in stratification showing good agreement with previous experiments. For the submarine simulations they considered two stratification environments: a sharp pycnocline at the shaft of the propeller and a pycnocline beneath the hull, finding that the generation of internal gravity waves increased resistance. With a mesh size of 31 million cells, their results showed near–field wake profiles to 12% of the body length downstream. In the same year, Norrison et al. [162] used a wall–modeled LES approach to simulate the self–propelled Joubert operating at a ship–scale Reynolds number for two different propellers. With roughly a 300
2.1. Low-Speed, Turbulent Wake Physics

million cell mesh, they resolved the near-field wake to 11% of the body length downstream. Petterson et al. [180] recently extended these computations to include in the Joubert in $\pm10^\circ$ yaw conditions. Posa and Balaras [182, 183] used the IB method and LES with a fully-resolved boundary layer to simulate the towed and self-propelled SUBOFF geometry at model scale. With a mesh size of 3.5 billion cells, they detailed the dramatic effect of stern–propeller interactions on the near-field flow. Wall–resolved LES appears to provide the highest fidelity but at a great computational cost.

2.1.4 Future Prognosis

The future of researching wake physics will continue to see advances in experimentation and simulation. Ship-scale experimentation will inevitably yield useful ground-truth data, which will be used to validate computational techniques. For example, a growing number of open-literature data campaigns are providing extensive oceanographic data. Following the predictions of Slotnick et al. [212], RANS will see less use in simulations that require highly-accurate turbulence modeling. Hybrid RANS/LES will provide a desirable tradeoff between accuracy and cost, and LES will see increasing use as computational power increases. With improving computational capabilities, one ever-present challenge is identifying fruitful areas of future research.

2.1.5 Conclusions

The literature was reviewed to identify low-speed, turbulent wake experiments and the current state-of-the-art in simulation. A large number of experiments have been conducted for a variety of geometries of or related to notional marine vehicles. These vehicles are observed in towed and self-propelled configurations, with and without stratified environments. Originally simple disks, spheres, or ellipsoids, most recent experiments included realistic, appended geometries. The use of simulation has expanded the types of cases that can be studied. While, DNS has seen success in identifying fundamental physics of stratified flows, lower fidelity methods are required to simulate realistic, operating vehicles. RANS computations provide adequate performance at a low cost, but hybrid RANS/LES and LES methods can better-resolve the genesis and interaction of vortex structures in the near-field wake.

Summary points:

1. Model-scale experiments have provided fundamental insight into low-speed, turbulent wakes, however there is a lack of ship scale data for validation and understanding of scaling effects.

2. Simulation extends the range of cases that can be studied.

3. Direct numerical simulation provides invaluable insight into fundamental flow physics.
4. Wall–resolved LES provides the highest fidelity near–field wake simulation of a realistic vehicle.

**Future issues:**

1. Provide useful validation data so that simulations can confidently predict fluid flows in cases difficult to experiment on.
2. Identify areas research important in the future.
3. Continue to improve simulation fidelity of near–field turbulence.
4. Efficiently leverage LES and hybrid RANS/LES for high fidelity, near–field simulations.

### 2.2 Vortex Breakdown

The present review examines research into the topic of vortex breakdown over the previous 60 years. The phenomenon is well studied, but a generalized explanation does not yet exist for all flows. Presented material is divided into a description of the phenomenon, followed by experimental and computational progress, with underlying advances in the theoretical understanding of vortex breakdown. A survey of state–of–the–art tools is given with a prognosis for future advancement. Conclusions are drawn and the key points are summarized.

#### 2.2.1 Introduction

Generally, vortex breakdown is described as changes in the vortex structure caused by changes in the ratio of azimuthal to axial velocity components [124]. There exist a total of seven breakdown modes, but the two predominant forms are “bubble” and “spiral” as shown by the dye–in–water experiments in Figure 2.1 [124].

Vortex breakdown holds importance in a number of applications. In combustion, it is used to control mixing in boilers, burners, diesel engines and gas–turbine engines. In hydro–turbines, such as the Francis turbine, breakdown reduces the efficiency and causes hydro–acoustic resonance. Vortices are also a desirable flow feature of some finite lifting surfaces. For example, a Delta–wing incorporates leading–edge vortices, and this vortical flow improves lift and stability. In the case of air–traffic safety, the breakdown of wing–tip generated vortices is desired so that aircraft can travel closer together, and more aircraft may be processed through an airport in a given amount of time. Similarly, the propeller–driven hub and tip vortical breakdown plays a role in the near–wake evolution of a propeller–driven vehicle. Finally, breakdown occurs in devices such as the Ranque–Hilsch vortex tube, which separates gases by molecular weight in hot and cold streams.
2.2. Vortex Breakdown

Figure 2.1: Photographs of bubble (a) and spiral modes (b) of vortex breakdown from experiment of Leibovich [124].

In general there exist three spatial regimes in such a flow: [124]

1. The approach flow is approximately–irrotational except for vorticity at the core. Axial changes are gradual, and the flow is said to be “supercritical”.

2. The breakdown region occurs after an adverse pressure gradient and stagnation point with areas of flow reversal.

3. The downstream, “subcritical” flow has a new, turbulent vortex with an expanded core. The axial–velocity profile is similar to that of a drag wake.

Given the physical complexity, researchers try to predict and explain the breakdown with unifying factors. Explanations of the phenomenon are typically given by one or more of the three main categories: [86]

1. As an analogy to boundary–layer separation with flow stagnation.

2. As a consequence of hydrodynamic instability.

3. As a dependence on a critical state, or wave phenomenon.

Being an area of widespread research previous reviews have been given by Hall [86], Leibovich [124, 125], Stuart [231], Escudier [59], Althaus et al. [3], and Lucca-Negro and O’Doherty [139].

Although the topic has been extensively studied for the previous 60 years, much still remains to be discovered regarding vortex breakdown. The three main explanations: analogy to boundary–layer separation, instability, and criticality each propose useful metrics in the prediction of breakdown in particular flows, but none can fully describe the phenomenon
for all cases. Largely a topic of experimental examination in the past, improving, high-fidelity numerical methods may yet shed light on a generalized, fundamental nature of the breakdown of vortices.

The present review is divided into three primary sections. First, the progress through experiments will be considered. Computational studies are discussed next with a review of advances in the theoretical understanding of vortex breakdown. Finally, the state-of-the-art research tools are described along with a prognosis for future advancement. Conclusions and summary points are given.

### 2.2.2 Experimental Advancement

Experimentation began in 1957, when Peckham and Atkinson [178] published the first observed breakdown of leading-edge vortices over a “delta” wing with high sweep at a high angle-of-attack. Following their setup, Lambourne and Bryer [122] showed that the observed spiral-structure frequency is a function of flow rate over a delta wing, and published the famous photograph in Figure 2.2 which shows simultaneously a bubble- and spiral-type breakdown occurring over the wing. In a departure from the delta wing setup, Harvey [90] was the first to show breakdown in a swirl-tube with guide-vanes. This setup removed the asymmetries of leading-edge vortices. He discovered flow reversal downstream of the breakdown suggesting the existence of a critical state. The breakdown region was compared to the intrusion of an elongated-spherical body in the flow forcing fluid around the boundary. Harvey [90] also noted that breakdown was highly sensitive to external influences.

![Figure 2.2: Photograph of delta-wing setup from Lambourne and Bryer [122].](image)

Examining measurements from a pressure cell and a hot-film anemometer, Cassidy and Falvey [31] found that the flow frequency is independent of the viscosity at high Reynolds numbers $Re$ and is constant for a specific $\Omega_v D / \rho Q^2$, where $\Omega_v$ is vortex core rotation rate, $D$ is diameter, $\rho$ is density, and $Q$ is volumetric flow rate. Furthermore, an upstream movement
of the breakdown was noted if any probe was present. Consequently, a continuous trend in
the design of experiments was transitioning from intrusive to non–intrusive probing such
as Laser Doppler Anemometry (LDA), Particle Tracking Velocimetry (PTV), and Particle
Image Velocimetry (PIV) methods [139].

In 1971, Sarpkaya [197, 198] observed three breakdown modes in a slightly diverging tube
with guide–vanes and dye in the water. By varying the flow–rate and circulation the three
modes found were: double–helix, spiral, and axisymmetric–bubble. The double–helix break-
down is show in Figure 2.3. So, breakdown–type can be considered a function of Reynolds
number Re and Circulation number \( \Omega \). Circulation number is defined as \( \Omega = \Gamma/ud \) where
\( \Gamma \) is the local circulation, \( u \) is the axial velocity, and \( D \) is the diameter. Swirl number \( S \) is
often used alternatively, being the ratio of the flux of axial to angular momentum. Using a
similar approach, Leibovich [124] would observe all seven of the known breakdown modes,
where types 3 through 6 were the least common and found at lower Re. These seven modes are:

- **Type 0**: axisymmetric–bubble.
- **Type 1**: asymmetric–bubble.
- **Type 2**: spiral.
- **Type 3**: combination of flattened–bubble (type 4) and spiral (type 2).
- **Type 4**: flattened–bubble.
- **Type 5**: double–helix.
- **Type 6**: central filament off center–axis at constant azimuth.

At higher Re, increases in Re and \( \Omega \) will generally move the breakdown upstream and lead to
transition from spiral to bubble forms of breakdown. In transitional regimes, the unsteady
flow may switch back and forth between these two forms [197].

![Photographs of double–helix breakdown at two different Reynolds and swirl numbers from Sarpkaya [197].](image-url)

(a) \( U_0D_0/\nu = 1150, \ G/U_0D_0 = 3.0 \)  
(b) \( U_0D_0/\nu = 1700, \ G/U_0D_0 = 2.3 \)
Introducing the slit–inlet swirl–tube setup with LDA measurements, Escudier et al. [60] observed the dramatic effect of exit contraction on the structure of the vortex. With a reduced exit diameter, axial and swirl velocity profiles sharpened, yielding higher velocity magnitudes and gradients. Escudier [58] also observed the bubble breakdown in a cylindrical tank with a rotating end–wall (CTRE) setup. Breakdown was described solely by the height–to–radius ratio of the cylinder and rotation Reynolds number. Using PTV, Brücker and Althaus [23, 24, 25] observed similarities between bubble– and spiral–form breakdowns suggesting that bubble–breakdowns were simply condensed spiral–breakdowns. Devenport et al. [52] investigated the evolution of a wing–tip vortex showing the development of self–similarity. Studying imperfections in the CTRE setup, Thompson and Hourigan [240] suggested that these experiments are more comparable to open–air vortices than previously thought. Looking at hydro–turbine efficiency, Susan-Resiga et al. [233] used LDA measurements to examine the runner outlet of a Francis tube, and developed an analytical method for representing the flow as a superposition of three vortices. Additionally, it was shown that a critical state was reached in the draft tube, something to be avoided in future designs to reduce hydro–acoustic resonance and improve pressure recovery. In this same turbine, Iliescu et al. [103] examined PIV measurements of the cavitating vortex rope within the draft tube to help offer a physical understanding of the phenomenon. Taking PIV measurements of a vertical swirling water jet, Mourtazin and Cohen [157] demonstrated the effect of buoyancy on vortex breakdown. When there is a negative/positive temperature difference between the core and surroundings the breakdown can be suppressed/enhanced respectively. So, “warm” vortices will breakdown sooner, whereas “cold” vortices will breakdown later. Taking stereoscopic PIV measurements in a CTRE, Lo Jacono et al. [138] showed that vortex breakdown could be controlled by the addition of vorticity source terms in the form of small rotating rods. Using a high aspect ratio CTRE, Sørensen et al. [217] showed the appearance of helical breakdown modes when only bubble modes had been observed previously in this setup. Recently, Oberleithner et al. [164] examined, in detail, the transition from supercritical to subcritical flow using PIV measurements of a turbulent swirling jet. Using time–resolved visualizations and velocimetry measurements of propeller wakes, Felli et al. [66] showed the effect of blade number and advance ratio on the propeller–blade hub and tip vortex breakdowns. Breakdown is delayed with decreasing blade number and increasing advance ratio.

2.2.3 Theoretical and Computational Advancement

Originally, available computational–power limited numerical study to incompressible, laminar, steady, and axisymmetric flow despite vortex breakdown being anything but in many cases [139]. In 1960 and 1962, Squire [229] and Benjamin [14, 15, 16] independently developed theories for an inviscid, steady, and axisymmetric vortex and the existence of a critical state. They both discovered that a swirl–ratio parameter defined as the ratio of maximum azimuthal velocity to maximum axial velocity $k = \max(V)/\max(U)$ could be used to describe the critical condition of a vortical flow. Flow was considered critical when
2.2. Vortex Breakdown

$k \geq 1.2$, thereby predicting an imminent breakdown. In 1967, Hall [87] used the failure of the quasi–cylindrical approximation (QCA) to predict breakdown, that is, the axisymmetric steady Navier–Stokes equations were solved. When the computed radial velocities approached infinite and a singularity was found in the solution, breakdown was expected to occur at that axial location. Results showed that breakdown depended on the degree of swirl and adverse pressure gradient in the same way as boundary–layer separation, showing an analog to boundary–layer theory. Aside from the assumptions, one major limitation of this technique is that it fails to describe the actual mechanics of vortex breakdown and the flow beyond. Hall [87] also noted two components of the axial pressure gradient: one from swirl and another from an external pressure gradient, showing the importance of boundary conditions in numerical and experimental setups.

Other than the analogy to boundary–layer separation, the explanations of criticality and instability were also considered. Randall and Leibovich [185] introduced concept of wave trapping. Wave propagation may be limited if dissipative and amplification effects are in equilibrium, thereby trapping the wave. The conditions for this to occur also allow a stagnation point to form. Escudier et al. [61] suggested link between criticality and instability. They argued that downstream of a breakdown the flow was unstable to “spiral disturbances”, and these disturbances were responsible for the particular breakdown type. So, while criticality may lead to breakdown, instability would determine the type. Shi and Shan [204] identified a link between stagnation and instability conditions, by showing a critical state when the QCA failed. Spall and Gatski [222] utilizing fully 3D, unsteady CFD, demonstrated that nearing breakdown, vorticity was redistributed from axial to radial and azimuthal directions. Spall et al. [224] proposed a criterion for breakdown based on the Rossby number $Ro = u/r\Omega_c$ in which $u$ is the axial velocity, $r$ the vortex core radius, and $\Omega_c$ the vortex core rotation rate. Breakdown would occur under the approximate critical condition $Ro \leq 0.65$ for $Re_{core} > 100$. Brown and Lopez [22] found that flow reversal could be predicted based on the upstream angle between vorticity and velocity vectors. More succinctly, breakdown could be predicted based on the production of negative azimuthal vorticity. Further advance required improvement to model fidelity.

Spall and Gatski [223] introduced turbulence into numerical models by using the $k – \epsilon$ and algebraic stress models. Upstream of the breakdown, the axial vorticity decayed more slowly in the turbulent case. Sotiropoulos and Ventikos [218, 219, 220] studied the CTRE using unsteady, laminar CFD to clarify the origin and mechanisms of the underlying physics. Ruith et al. [195] implemented axisymmetric Direct Numerical Simulation (DNS) to show how several breakdown modes are based on a wave number. Gallaire et al. [74] expanded on this work interpreting the spiral breakdown as a global, nonlinear mode that arises from an axisymmetric breakdown state, similar to the unstable wake behind a bluff body. Behera et al. [13] used CFD with the RNG $k – \epsilon$ turbulence model and experimental validation to optimize geometric properties of the Ranque–Hilsch vortex tube to obtain the maximum hot gas and minimum cold gas temperatures.

Many recent studies focus on practical applications such as hydro–turbine technology. Study-
ing the Francis turbine, Susan-Resiga et al. [234] used 3D unsteady CFD with the $k – \epsilon$ turbulence model to show how the addition of a center jet in the vortical draft–tube inflow could mitigate the accompanying precessing vortex rope to eliminate pressure fluctuations and reduce hydro–acoustic resonance. The jet would use flow redirected from the spiral casing through the center shaft and out of the runner crown tip, imparting momentum to the center of the swirling vortex. Susan-Resiga et al. [235] built upon this work using a Reynolds stress model (RSM) to show that, with the aid of this jet, breakdown can be eliminated entirely and losses minimized.

\subsection{2.2.4 State–of–the–Art Tools and Future Advancement}

At present time, several state–of–the–art tools exist for the study of vortex breakdown. The first tool is experimentation such as Oberleithner et al. [163] using Proper Orthogonal Decomposition (POD) methods on measured PIV data. By reconstructing time–periodic 3D velocity data from 2D snapshots, the results suggested the existence of a singular–frequency, global breakdown mode arising from self–excitation. Beyond experiment, computational techniques are increasing in value because of the cost–effective ability to run simulations for specific case studies. One example is the use of Direct Numerical Simulation (DNS) by Ruith et al. [196], who implemented DNS for 3D, incompressible, spatially and temporally evolving, swirling laminar jets, and explored the proper implementation of boundary conditions. Due to the expensive nature of DNS, present studies are limited to low Re. To look at 3D fields at higher Re, turbulence can be modeled using a closure method. Hybrid RANS/LES techniques are popular because they combine the strengths of Reynolds–averaging at near–wall boundaries and Large Eddy Simulation (LES) to simulate swirling eddies elsewhere [221]. In 2014, Foroutan and Yavuzkurt [71] used one such technique known as the Partially–Averaged Navier–Stokes (PANS) method to study swirling flow in two cases: through an abrupt expansion and in the Francis–turbine draft tube. They showed improved results over Delayed–Detached Eddy Simulation (DDES) and $k – \omega$ SST models. Their improved PANS method, modeled the unresolved–to–total turbulent kinetic energy ratio parameter $f_k$ by integrating over analytic turbulent kinetic energy spectrum.

Given the present state–of–the–art, the prognosis for future advancement is clear. Experiments will continue to be valuable because of the “ground truth” that they provide. That is, while error is still present, it is generally quantifiable, and the model–form uncertainty that arises in numerical approaches is not a factor. DNS will be valuable because of similar reasoning. Although expensive, this method of simulation provides the arguably highest fidelity model of general fluid simulation. As computing power grows, fully 3D vortex breakdown will be examined through DNS. Until such capabilities exist, however, the next best approach may be hybrid RANS/LES CFD, because it provides a strong physical model of turbulence at an affordable cost. Regardless of the method–of–study used, the control of vortex breakdown will likely be a growing field–of–interest because of how common the phenomenon is in practical applications.
2.3. Set–Based Design

2.2.5 Conclusions

In this review, the breakdown of vortical flows was examined. A fundamental explanation of the phenomenon was presented, followed by a historical review of experimental, computational, and theoretical advancements. State–of–the–Art tools were considered along with the prognosis for future advancement. Key summary points and future issues are listed.

Summary points:

1. Vortex breakdown can be described as a change in vortex structure caused by a change in the ratio of axial to azimuthal velocity components.

2. There exist seven breakdown modes with the two predominant being the bubble– and spiral–forms.

3. There exist three spatial regimes: the approximately–irrotational, supercritical upstream flow, the breakdown region with flow reversal and stagnation, and the subcritical downstream that resembles a turbulent drag wake.

4. Vortex breakdown is explained through three phenomena: an analogy to a boundary layer, a consequence of hydrodynamic instability, and a dependence on a critical state.

Future issues:

1. Experimentation will continue to be valuable.

2. Fully 3D DNS will offer new insight into the phenomenon when computation becomes affordable.

3. Presently, the hybrid RANS/LES approaches will offer an affordable simulation approach, especially when vortex breakdown is just a single component of a larger problem.

4. The control of vortex breakdown will be an active area of interest due to the numerous applications.

2.3 Set–Based Design

Traditional point–based design methods converge on a singular deterministic design point. Because of poor initial approximations, unforeseen issues, and changes in requirements, these methods often result in sub–optimal or even inadequate designs that require costly rework. Within the last 30 years, a new paradigm has emerged known as set–based design. Instead of viewing individual designs with deterministic parameters, sets of alternatives are considered.
Ranges or even distributions of parameters compose a set of designs, and the intersection of multiple sets create the feasible design space. Infeasible designs are eliminated as the design process progresses, analysis methods improve in detail, and more information is discovered. The methodology provides flexible designs and resiliency towards uncertainty. In this review, the origin, history, and advances of set–based design are surveyed. Comparison is made to the traditional point–based design methodology, and the inclusion of set–based design in multidisciplinary design optimization is considered. Finally, a prognosis for the future is made and summary points are given.

2.3.1 Introduction

The difficulty of engineering design is well–understood, especially by those who pursue the end–to–end, concurrent process. Engineering design is an open–ended problem in which many paths can be taken to reach the final product. How can the design team most–easily reach the best solution? There is no clear answer because the decision–making process itself is a design problem with high variability and uncertainty that depend on each individual case. Experience and careful deliberation generally guide the design process. The traditional design process is often referred to as point–based design (PBD), in which singular designs are evaluated and iterated upon. Within the last 30 years, a “new” paradigm has emerged known as set–based design (SBD), or equivalently, set–based concurrent engineering, in which sets of designs are considered. SBD, succinctly, requires the design to remain flexible as long as possible so that parameters may become increasingly constrained as respective disciplinary experts accumulate more knowledge throughout the process. In this way poor initial approximations, unforeseen issues, and changes in requirements are more–easily addressed and a better, more–robust design can be converged upon.

SBD has proven successful in the automotive industry, shows great popularity in naval engineering, and has expanded to many other fields. It is most–promising for problems that demand novel, innovative designs, whereas PBD is better–suited for incremental design problems such as altering an existing design to meet new requirements. Although multi–disciplinary design optimization (MDO) is typically considered a PBD method, the process can be modified to follow the SBD paradigm. The greatest criticism of SBD is its difficulty in implementation, however, advances in the literature continue to enhance the process to a broader range of problems. Because of its capacity to lower costs and provide more–resilient designs despite large uncertainties, SBD will continue to grow as a powerful concurrent engineering design paradigm.

In this review, the history and merits of SBD will be discussed within the context of concurrent engineering design. The origin and growth of SBD will be explained in terms of both formulation and applications. Observations will be drawn between SBD and traditional PBD methods. Inclusion of SBD into MDO will be considered. The growing interest in naval applications will be discussed, as well as successful applications in other fields. Finally,
2.3. Set–Based Design

a prognosis for the future will be given.

2.3.2 Origin and History

Ward and Seering [253, 254] demonstrated set–based concepts in the early 1990s when developing a computer program to design mechanical and hydraulic power transmission systems. This computer program would take in ranges of design requirements from the user, search a library of standard catalog components, and return all subsystems that could fulfill the requirements. Infeasible subsystems would be eliminated, leaving behind feasible sets of designs, and the component catalog numbers that would form each subsystem. The term “set–based concurrent engineering”, however, was first coined by Ward et al. [251, 252] when describing successful practices of Toyota Motor Corporation in the automotive industry. Ward et al. described how Toyota would produce cars quickly and cheaply by purposely communicating ambiguously, delaying decisions, and considering large numbers of prototypes. While these choices might seem to be undesirable in the view of traditional PBD, they are the underlying attributes that make SBD successful. Liker et al. [131] found that set–based practices were more prevalent among Japanese automotive part suppliers than their U.S. counterparts, distinguishing between set–based and point–based approaches. Sobek II [216] distilled Toyota’s success with SBD into five essential aspects: solution space exploration and set–narrowing, design space mapping, designing for a set of environments, set–based communication, and set–based concurrent engineering. Sobek et al. [215] would later describe SBD through just three principles: mapping of the design space, integration by intersection, and the establishment of feasibility prior to commitment. With the demonstrated success of SBD in the automotive industry researchers looked to improve general SBD concepts and apply the paradigm to other applications.

Finch and Ward [68] provided methods for eliminating infeasible designs and applies it to the SBD of a simple electronic circuit, and Finch [67] showed the benefits of interval computation in design processes with large uncertainty. In 1998 Bernstein [18] surveyed the usage of SBD in the aerospace industry by characterizing SBD by two main principles: first, that designers should consider a large number of alternatives that are gradually narrowed, and second, that the disciplinary experts should independently generate these sets from their own perspective and look for regions of intersection between them to establish the feasible design space. While the aerospace companies used some set–based concepts, none fully matched these criteria. Bernstein therefore proposed a model to implement SBD, taking concepts from lean manufacturing. Parunak et al. [177] introduced market forces into the SBD process through the Responsible Agents for Product-Process Integrated Design (RAPPID) project [175]. Disciplinary experts would act as agents in a marketplace, buying and selling components of the design [174, 176]. Applying this concept to the design of a hatch–covered, cellular, container ship, Parsons et al. [173] showed that RAPPID improves three core problems in design: planning, coupling, and prioritizing. Parsons and Singer [172], Singer [206], Wang and Terpenny [249] incorporated fuzzy logic–based communication
system to better express agent desires and improve design convergence. Nahm and Ishikawa [158] created a method with the use of interval arithmetic to better–formulate designer preference in both the design and performance space. Ford and Sobek [70] highlighted the merits of delaying alternative selection and demonstrated that converging too quickly or slowly decreases value in a concurrent engineering project. Malak Jr and Paredis developed a construct called parameterized efficient sets to formalize tradeoff–space relationships [143, 144]. Malak et al. [142] went on to build a framework which combines multi–attribute utility theory with SBD. Shahan and Seepersad [203] showed how Bayesian networks can be used to express arbitrary regions of the design space with the potential to capture a designer’s preference, through which Pareto optimal solutions are found. In the late 2000s growing interest appeared for SBD in United States Naval applications.

A 2008 letter from Vice Admiral Paul Sullivan, Commander of the Naval Sea Systems Command, detailed the need to improve design and analysis tools and to make them compatible with SBD practices [232]. The desire to implement SBD in the naval design was reiterated in the following years [54, 56, 117]. Singer et al. [205] described the SBD process and how it could replace traditional PBD methods in naval ship design to provide more–robust concepts in the presence of changing requirements and design discoveries. SBD would soon be applied to submarine design [72, 171], ship design [32, 57, 82, 88, 117, 146, 148, 149, 151, 193], power systems [241, 242], fleet design [1], and even the U.S. Department of Defense acquisition process [33, 107].

More recent advancements have been made in the 2010s. Avigad and Moshaiov [10, 11] developed a computational approach to solve multi–objective set–based problems. The use of fuzzy–logic in defining and analyzing sets was extended [81, 83, 147]. McKenney [150] focused on the issue of set reduction in the presence of changing design requirements, which, despite the inherent resiliency of SBD, can still be a point of failure. Using the Markov Decision Process to narrow sets, more robust decision paths were identified. Yannou et al. [259] proposed narrowing sets based on coverage of end–product usage scenarios. Ghosh and Seering [77] reviewed SBD literature and identified two core set–based thinking principles: delay decision making and concurrently consider sets of distinct alternatives. Citing the benefits of reusing physical parts in life–cycle costs of platforms, Levandowski [126] examined the more abstract reuse of technologies, requirements, and concepts. Levandowski used configurable system elements to relate platforms between lifecycles in the SBD process to design long–term efficient designs. Hannapel and Vlahopoulos [88], and later Rapp et al. [187], examined the use of SBD in multidisciplinary design optimization (MDO), which is traditionally considered a PBD method. Kennedy et al. [118] demonstrated how to best–apply SBD to reduce rework in a general systems engineering environment. Gray et al. [82] compared SBD to PBD through an experiment. Using the respective paradigms, two separate teams solved the same ship–design problem with two mid–design changes in requirements to test robustness. SBD was found to be cheaper, lower–risk, and more robust. Ross [193] and Hartman [89] separately developed new methodologies to estimate resilience in the design space, while Rapp et al. [186] worked to proactively estimate the cost of mid–development
changes in design. Expanding SBD to passenger aircraft family design, Riaz et al. [191] showed that SBD is best-suited to non-conventional design problems requiring innovation, as opposed to PBD, which is better-suited to conventional problems involving incremental improvement. Finally, Specking et al. [225] reviewed the literature and argued the benefits of applying tradeoff analytics to more-rigorously evaluate the design space.

### 2.3.3 Set–Based vs. Point–Based Design

Emerging within the last 30 years, set–based concepts are relatively recent as compared to conventional point–based concepts in design. Both have unique strengths and weaknesses. The following discussion will describe both paradigms and then compare the two. A discussion of the suitability of multidisciplinary design optimization (MDO) in SBD will follow.

**Point–Based Design**

The traditional point–based approach to design is deterministic and iterative. Typically concepts are formulated and compared to one–another using low–fidelity methods. After studying initial tradeoffs, design parameters are quickly solidified so that other disciplinary leads can deterministically analyze and select their design parameters. The process can be visualized with the design spiral diagram for a marine vehicle shown in Figure 2.4. This notional design spiral follows the 1959 criteria detailed by Evans [63]; a more recent diagram can be found in the work of Nordin [161] for the design of a submarine. A chain of dependence is established between disciplines in the analysis of the design. So, the final design for a particular concept is converged upon iteratively. As the design progresses, higher–expense and higher–fidelity methods are used to evaluate performance. With growing detail in the design and better accuracy in simulation, design flaws are discovered that must be addressed. Unfortunately, changes often force the new design to be sub–optimal or entirely infeasible requiring a complete rework in the initial concept. Additionally, if the design requirements are modified mid–process, the point design may require a radical rework. Uncertainty also lies in the lifetime application of the product since “mission” requirements may transform as real–world necessities change. PBD lacks robustness to these external uncertainties that are hard or impossible to predict during the design process.

Liker et al. [131] lists the following five basic steps of PBD.

1. Problem is defined.
2. Many concepts are generated.
3. Preliminary analysis of the concepts leads to a single concept to proceed with.
4. The concept is studied further with higher fidelity methods and modified until all requirements are met.
5. Restart from step 1 or 2 if the concept fails to meet the requirements.

Figure 2.4: Classical point–based design spiral of a marine vehicle following the criteria of Evans [63].

**Set–Based Design**

Instead of viewing a concept or parameter as singular and deterministic, SBD requires communicating in ranges or distributions. Feasible and infeasible regions of the design space are mapped. Each engineering discipline can formulate sets from their own perspective. By comparing the intersections of these multi–disciplinary sets, a robust design space is discovered. As modeling fidelity is improved, the sets are gradually narrowed until the design is finalized. The narrowing of sets can be considered parallel to the Method of Controlled Convergence, where only Pareto optimal designs are kept and Pareto dominated designs are discarded [205]. The SBD process is described by Bernstein [18] and illustrated in Figure 2.5 for a generic design process involving three disciplines.

1. First, the separate disciplines determine feasible sets over the entire design space from their own perspective.
2. Next, the disciplines expand their parameters to establish regions of overlap.
3. Working together, they expand the region of overlap.
4. As analysis methods improve and more information is discovered about the design, the overlap region shrinks.

5. The solution space is further-narrowed until a singular design is converged upon.

SBD works well because work is front-loaded to the early conceptual phase when uncertainty is largest, information is scarce, and decisions have the greatest impact. Bernstein [18] illustrates the discrepancy in committed and incurred costs during a typical PBD product development cycle. Early in development, committed costs are high while incurred costs are still low. Stakeholders must commit resources given the evaluation of the designers who are operating under a high degree of uncertainty. As the project progresses, knowledge improves, but the influence of the stakeholders decreases, because if requirements are changed, modifications to the design are costly. Figure 2.6 illustrates the effect of SBD on the product development cycle. Notional curves follow the portrayal of Bernstein [18]. Knowledge rapidly increases through the early stages of the product development, and committed costs follow suit. Management influence decreases over time. SBD shifts the committed costs and management influence curves. Because decisions are delayed and the design is left open longer, committed costs can be lowered, while the influence of stakeholders to change the design increases as knowledge is gained. The end effect is a more-resilient design. The fundamental principles of SBD can be summarized as follows [18, 205, 215].

1. **Map the design space.** Considering a large number of designs to establish feasible regions and explore tradeoffs. Allow specialists to evaluate design from their own perspectives.
2. *Integrate by intersection.* Identify regions of intersection to establish feasible sets with conceptual robustness.

3. *Establish feasibility prior to commitment.* Gradually narrow sets as modeling detail improves.

Figure 2.6: Effect of SBD on product development cycle with notional curves inspired by Bernstein [18].

**Comparison Between Design Paradigms**

Several studies have attempted to directly compare SBD and PBD. Madhavan et al. [141] showed that SBD could reduce costly iterations between design teams present in their previous PBD methodology when designing a downhole module for an oilfield. Gray et al. [82] formulated a ship combatant design problem that would be completed by two separate design teams: one using PBD and the other, SBD. Design requirements were changed twice in the middle of the process to test the robustness of the methods. At the end of the process, SBD was found to be cheaper, lower-risk, and more robust. Riaz et al. [191] applied both SBD and PBD to the design a three-member aircraft family.

Riaz et al. ultimately found that SBD performs best for non-conventional design problems that allow for innovative solutions. PBD was better-suited for design problems that required incremental improvement of existing platforms, where empirical data is available. Bernstein [18] provides similar suggestions regarding the decision between using SBD or PBD. SBD should be used when the development project can be characterized by: a large number of
Design variables, a high degree of variable coupling, conflicting or flexible requirements, or poorly understood technologies or design problems. PBD should be used when the development project is characterized by: requirements for specific technologies, a small number of design parameters, or well-understood technologies or design problems.

In the design of small Unmanned Aerial Vehicles (UAV) for surveillance missions, Small et al. [213, 214] showed that SBD can better map the design trade-space and identify the Pareto frontier, whereas a PBD approach yielded sub-optimal solutions that failed to resolve the Pareto curve. A major criticism of SBD is that it is difficult to implement for practical problems, however as the literature continues to grow this issue will diminish. The techniques for implementation continue to expand, and the breadth of applications is increasing as more industries adopt set-based practices.

Relationship to Multidisciplinary Design Optimization

MDO seeks to find a singular, optimal design point in a multi-dimensional parameter space. Because the end result of a converged optimization is typically a single point design, MDO is traditionally classified as a PBD method. So, MDO and SBD concepts are often seen as mutually exclusive. However, with advances in the analytical formulation of SBD [10, 11, 141, 158, 203, 249], researchers have extended set-based concepts into the MDO formulation. Hannapel and Vlahopoulos [88] developed a new MDO algorithm which utilized sets of design variable values rather than singular points. This MDO routine narrows the sets of design variables such that the overall performance improves. The end result is a high-performing, feasible design space rather than a singular point. Hannapel and Vlahopoulos applied their MDO routine to the design of a ship, where resistance, maneuvering, seakeeping, and estimated cost were considered as the design criteria. After completing their SBD-MDO routine, they performed a point-based MDO routine on the resulting design space and found that their final design performed better than one that followed the point-based MDO routine from the start. Rapp et al. [187] developed a mathematical framework to synthesize performance from competing designs into an optimality structure. The so-called “Contribution-to-Design” function can then be optimized, providing resilient design sets in the presence of uncertainty [186]. Current analytical frameworks allow for future advances in combined SBD-MDO approaches, which can provide benefits of both of the approaches.

2.3.4 Practical Applications

SBD has been applied to many fields of engineering. Originally found within the practices of the Japanese automotive industry, which has been well-documented [70, 77, 215, 216, 251, 252, 253, 254], set-based thinking expanded to many other applications. SBD is most popular in naval engineering due to a push by the U.S. Navy, but it has also seen success in a variety of other fields.
Naval Engineering

At the direction of Vice Admiral Paul Sullivan in 2008, SBD research was applied to United States Naval engineering [205, 232]. Frye [72] explored how to replace point–based methodology with set–based concepts in naval ship design. Frye first studied SBD in the Ship–to–Shore–Connector (SSC) program, which involves the design and acquisition of a future U.S. Navy Air Cushion Vehicle that can transport personnel and equipment from ship–to–shore. Taking insights from that framework, Frye examined SBD in the early concept exploration of the Ohio–class submarine replacement. Mebane et al. [151] would further study SBD in the SSC program, citing methodology, challenges, and future insight into the process. Based on insights from the SSC program, McKenney and Singer [148] worked to improve design–space exploration by developing an interactive software tool, after previously examining the general application of SBD to naval ships [149]. Gray [83] studied representations of uncertainty in the SBD of ships. Gray and Singer [81] would develop a tool for set–based ship design that facilitated designer preference and communication, and argued the desirability of SBD over PBD [82]. Chalfant [32] provided a tutorial in the conceptual design of an electric ship including, among other methodologies, SBD. Adopting set–based concepts, McCauley et al. [146] formulated a method to design flexible ship architectures. Several researchers have studied SBD in the U.S. Department of Defense acquisition process [33, 107]. Alessandria et al. [1] considered fleet architecture for the U.S. naval force, designing for robust ships and platforms. Ehlies [57] studied the future design of light aircraft carriers. Parker et al. [171] provided a strategy for applying SBD to the future SSNX submarine, considering capabilities, technologies, historical platforms, and programmatic considerations. Ross [193] studied the Littoral Combat Ship, examining existing, modified, and future designs while looking for robustness to meet future needs. Toshon et al. [241, 242] examined all–electric shipboard power systems, developing designs robust to uncontrollable noise factors. Most recently, Hunt et al. [99] created the Set–Based Design Evaluation Tool (SET), a software aimed to aid in naval systems engineering by allowing users to easily understand a complex, interdependent design space and make informed decisions.

Outside of the U.S. Navy, other researchers have applied SBD to naval engineering. Gaspar et al. [75] applied set–based concepts to the systems engineering of ships, identifying five aspects of complexity: structural, behavioral, contextual, temporal, and perceptual. Hannapel and Vlahopoulos [88] implemented SBD–MDO into ship design. Milanovic [156] applied SBD to Norwegian ship–building projects. Royset et al. [194] examined the design of a high–speed, super–cavitating hydrofoil, generating designs that were low–risk and uncomplicated.

Other Applications

SBD research has seen interest in many fields outside of naval engineering. Multiple researchers have worked to provide SBD analysis and feedback capabilities in Computer–Aided Design (CAD) tools [105, 120, 159]. Madhavan et al. [141] studied the industrial design of a
downhole module for an oilfield, finding that SBD could provide advantages over their previous PBD strategy by reducing costly iterations between design teams. Raudberget [188] examined sensor selection in an industrial setting, showing that SBD outperformed Pugh’s method [184]. De Leon Martin and Keles [47] included SBD in manufacturing production planning. Lee et al. [123] incorporated set–based concepts into the design of high–rise buildings, extending the framework to improve efficiency, safety, constructability, and economic viability. Inoue et al. [106] applied SBD to the automotive design of a structural frame. Yannou et al. [259] applied SBD to a family of jigsaws, demonstrating the advantages of narrowing sets based on coverage of end–product usage scenarios. Unal and Warn [244] showed how SBD could be used to help guide infrastructure recovery after a natural disaster. Riaz et al. [191] studied the SBD of an aircraft family design, demonstrating the advantages of SBD over traditional PBD approaches. Ammar et al. [4] examined the industrial mechatronic design of an Eletronic Throttle Body. Hartman [89] compared cost and utility for numerous alternatives of military systems. Lycke [140] studied the incorporation of SBD in Norwegian product manufacturing. Schmidt et al. [201] looked to restructure the software–engineering repository tree to include set–based concepts. Finally, Small et al. [213, 214] studied the design of a small Unmanned Aerial Vehicle (UAV) for surveillance missions, illustrating the advantages of SBD over PBD.

2.3.5 Future Prognosis

Because of its proven value in the automotive industry, naval engineering, and many other design applications, the principles of set–based thinking will continue to play an important role in engineering design. However, because of perceived difficulties in implementation, additional extensions to framework and contributions to the literature will be important. SBD concepts will be most–valuable to problems with large uncertainty, whether in the product development phase or throughout the operational life cycle of the end–product. The inclusion of SBD in MDO shows promise because the practical capabilities of MDO can be combined with the beneficial qualities of SBD. Based on recent history, SBD will likely see heavy use within naval engineering and increasingly diverse use in other applications.

2.3.6 Conclusions

This review fully explored the topic of set–based design in the literature. Beginning with its origin in the automotive industry, expansion to other industries, and the surge in popularity in naval engineering, the broad applicability of SBD was demonstrated. Historical extensions to analytical capabilities were also explored. SBD was described, compared, and contrasted to the classical point–based design approach, and the main principles of both were discussed. Traditionally formulated as a PBD method, MDO was shown to allow for the inclusion of SBD concepts. The breadth of successful SBD applications were listed, and finally, a
prognosis for the future of SBD was made. Summary points and future issues are listed.

Summary points:

1. A relatively recent design paradigm, SBD has shown a large surge in popularity, most–
   notably in naval engineering but also in a wide variety of industries.

2. PBD is better–suited for incremental designs, while SBD is better–suited for innovative
   designs, since it can provide resilient designs in the presence of large uncertainty.

3. Advancements to the analytical framework have allowed for better communication and
   representation of the design space, as well as inclusion into the MDO framework.

Future issues:

1. SBD will likely see heavy use within naval engineering and increasingly diverse use in
   other applications.

2. SBD concepts will be most–valuable to problems with large uncertainty.

3. Inclusion within the MDO process further potential for further advancement.
Bibliography


[34] Nathan Chase. Simulations of the darpa suboff submarine including self-propulsion with the e1619 propeller. Technical report, Iowa University, Iowa City, 2012.


[229] HB Squire. *Analysis of the vortex breakdown phenomenon*. Imperial College of Science and Technology, Aeronautics Department, 1960.


Chapter 3

Validation and Scaling of the Stratified Near Wake of a Self–Propelled Body

(Work adapted from Jones and Paterson [22].)

Abstract

The unsteady near wake of a self–propelled body in a linearly stratified environment is studied through the novel use of an actuator–line model for the propeller in unsteady, Reynolds–averaged Navier–Stokes simulations. A comparison to experiment at $Re_L = 1.6 \times 10^6$ shows agreement in wake profiles and the emergence of individual blade–wake structures. Increasing the Reynolds number to $Re_L = 310 \times 10^6$ reveals a delayed vortex breakdown and relative reduction in velocity profile magnitude, in addition to a thinner boundary layer and reduced skin–friction coefficient as predicted by theory. Flow is shown to be independent of internal Froude number. Downstream turbulent–kinetic, mean–kinetic, and potential energies reveal the persistence of swirl and growth of potential energy, which is embodied as a mixed patch. While velocity and turbulent–kinetic energy are axisymmetric in the mixed patch, the temperature field grows into a unique profile that is asymmetric in the azimuthal direction. Simple, analytical profiles are fit to the axisymmetric profiles. A new analytical method is developed to imitate the asymmetric temperature profile.
3.1 Introduction

Propeller wakes are known, through experiment, to evolve from the highly–complex near wake, in which distinct blade flow features are discernible, to the far wake, in which distinct blade flow features have blended together to form a mixed, nearly axisymmetric field [19, 20]. Sirviente and Patel [33] have shown that spatially, the near wake exists roughly within the first four diameters downstream of the propeller, \( x'/D_p < 4 \); the intermediate region within \( 4 < x'/D_p < 12 \); and the far wake somewhere beyond \( x'/D_p > 12 \). The evolution from near wake to far wake depends on flow parameters such as Reynolds number and Swirl number as well as configurations such as the propeller geometry and number of blades [15]. The rotating propeller is the driver of both the swirl and helical vortices that shed off of the root and tips of the blades into the near wake. Experiments have shown the physical contributions of swirl on various wakes [34], yet its role in the evolution from near to far wake is still largely unknown. Stratification itself is shown to play a major role in the development of propelled wakes [27]. Originally studied as a disc–with–center–jet [29] and later with self–propelled axisymmetric bodies [23, 31], the net–zero–momentum wake holds particular interest to researchers because of its usefulness as a theoretical model of a propeller–driven marine vehicle.

Beyond experiment, the study of propelled flows includes several methods that can be classified with increasing fidelity and computational expense [30].

1. Analytical: analytic profiles of a self–similar far wake are used as the starting condition in simulation. This method precludes the effects of individual propeller blades.

2. Blade Element Momentum (BEM) theory: two–dimensional blade elements satisfy global conservation of momentum to calculate aerodynamic blade characteristics.


4. Panel methods: similar to VLM, an inviscid flow field is prescribed over discrete panels. In this case, blade geometry is more accurately included.

5. Generalized actuator methods: Actuator Disk (AD), Actuator Line (AL), and Actuator Surface (AS) impose a body force over a region of volumes in a Computational Fluid Dynamics (CFD) simulation to impart the effects of propeller on the surrounding flow.

6. Fully resolved propeller: transient propeller geometry is directly built into the CFD mesh as an unsteady boundary condition.

Although the fully resolved propeller offers the most accurate simulation, its computational requirements are often very large. Actuator methods are easier to implement and more computationally efficient, thereby making them a popular choice in the application of simulating
turbines in wind farms [28]. The AL model, used presently, represents each propeller blade as a line of rotating body forces that follow the path of each propeller blade. While the AL model neglects viscous effects of the propeller boundary layer, it still captures the important details of propeller root and tip vortices [21]. Originally derived from the work of Sorensen and Shen [35], further developments to the AL model have been made by Troldborg et al. [37][36][38], and Sibuet Watters and Masson [32]. The present study uses the National Renewable Energy Laboratory (NREL) implementation known as Simulator For Wind Farm Applications (SOWFA) [11, 12]. This implementation uses the open-source C++ based CFD framework known as OpenFOAM (Open Field Operations and Manipulations) [1].

At present time, little is known about the interaction between stratification and swirl throughout the near-wake evolution. To unravel this interaction, propeller and body geometry must be considered together because of the key role they play on the mechanisms of wake development. Additional complexity arises in the prediction of vortex breakdown and Reynolds-number scaling effects. Although the stratification has only a small influence on the flow due to overpowering inertial forces, the flow field has significant influence on the thermal distribution. As will be shown, swirl and stratification alter the structure and flow properties of the wake profile particularly in the development of potential energy.

This study builds off of the work of Jones and Paterson [22]. The Navier–Stokes equations are solved to examine the near-wake evolution of a turbulent, net-zero-momentum propeller wake in the presence of thermal stratification typical of an oceanic environment. An AL model is used to simulate a rotating propeller attached to the hull of an axisymmetric body. To the author’s knowledge, this is the first known implementation of this kind. The axisymmetric Iowa Body geometry is chosen to allow for comparison to the experimental data of Hyun and Patel [20] for the non-stratified, experimental-model-scale case. This is the only known experiment with phase-averaged, mean-velocity, and Reynolds-stress data. The significance of dimensional parameters is studied through Buckingham’s II method and directly through simulation. Averaged circumferential flow parameters are examined in the near wake and intermediate region to investigate propeller-wake evolution under the influence of stratification and swirl, and the impact of swirl on potential energy. Comparisons are made to the theoretical disc-with-center-jet that is often used in computational studies [5, 9, 17]. Analytical curves are fit to the axisymmetric flow fields. The temperature deviation field is not symmetric, however, so a new analytical profile is developed to imitate the simulated mixed-patch data.
3.2 Approach

3.2.1 Governing Equations

The unsteady Reynolds–averaged Navier–Stokes (RANS) equations in Boussinesq form govern the flow field in these simulations. An additional body force term \( f_p \) is included to account for the propeller model.

\[
\frac{\partial U_j}{\partial x_j} = 0 \tag{3.1}
\]

\[
\frac{\partial U_i}{\partial t} + \frac{\partial (U_i U_j)}{\partial x_j} = -\frac{1}{\rho_0} \frac{\partial \rho}{\partial x_i} + \nu \frac{\partial^2 U_i}{\partial x_j \partial x_j} + \frac{\partial}{\partial x_j} u_i' u_j' + \frac{\Delta \rho}{\rho_0} g_j \delta_{ij} + \frac{1}{\rho_0} f_p \tag{3.2}
\]

The variables in these equations include the non–inertial velocity \( U_i \), time \( t \), kinematic viscosity \( \nu \), and density \( \rho \). Here, density is expressed as \( \rho = \rho_0 + \Delta \rho \), where \( \rho_0 \) is a reference value and \( \Delta \rho \) is the deviation from that value. The vertical position coordinate \( z \) is upward–positive, and the gravitational vector \( g_j \) points downward in the negative \( z \) direction. This formulation includes the piezometric pressure, \( \hat{\rho} = \rho - \rho_0 g z \) with \( g \) being the magnitude of the gravitational vector.

A custom solver written with the OpenFOAM CFD framework is used to solve the governing equations. The salinity \( T \) and \( S \) temperature transport equations are solved followed by the corresponding turbulent fluctuations. The \( T \) and \( S \) transport equations are,

\[
\frac{\partial T}{\partial t} + \frac{\partial (U_j T)}{\partial x_j} = \kappa_T \frac{\partial^2 T}{\partial x_j \partial x_j} + \frac{\partial}{\partial x_j} u_j' T' \tag{3.3}
\]

\[
\frac{\partial S}{\partial t} + \frac{\partial (U_j S)}{\partial x_j} = \kappa_S \frac{\partial^2 S}{\partial x_j \partial x_j} + \frac{\partial}{\partial x_j} u_j' S' \tag{3.4}
\]

where \( \kappa_T \) and \( \kappa_S \) are diffusion coefficients.

The Reynolds stresses \( u_i' u_j' \) and accompanying turbulent fluxes \( u_j' T' \) and \( u_j' S' \) are computed through a linear eddy–viscosity closure model.

\[
-\overline{u_i' u_j'} = 2 \nu_t S_{ij} - \frac{2}{3} k \delta_{ij} \tag{3.5}
\]

\[
-\overline{u_j' T'} = \frac{\nu_t}{\sigma_T} \overline{\frac{\partial T}{\partial x_j}} \tag{3.6}
\]

\[
-\overline{u_j' S'} = \frac{\nu_t}{\sigma_S} \overline{\frac{\partial S}{\partial x_j}} \tag{3.7}
\]

Turbulence may be modeled using a RANS or hybrid RANS Large Eddy Simulation (LES) approach. In this case the \( k–\omega \) SST turbulence model is chosen to compute the eddy viscosity \( \nu_t \) due to its ease of implementation and relative strength in computing an attached flow
over a body [26]. Production terms in the $k - \omega$ equations are modified to include buoyancy effects, but in the near wake they are small in comparison to the production due to shear.

Density is computed by solving the UNESCO seawater equation–of–state [16]. For the given problem, it is appropriate to approximate the secant bulk modulus as constant at sea–level, ambient conditions, even though it is a function of salinity, temperature, and pressure. Thus, the secant bulk modulus is

$$K(S, T, p) = K(0, 20, p_{atm})$$

where $p_{atm}$ is atmospheric pressure.

Additionally, substituting the hydrostatic pressure for the total pressure, the equation–of–state becomes,

$$\rho(S, T, p) = \frac{\rho(S, T, 0)}{1 - \frac{p}{K(S, T, p)}} \tag{3.8}$$

$$\rho(S, T, 0) = (a_0 + a_1 T + a_2 T^2 + a_3 T^3 + a_4 T^4 + a_5 T^5)$$

$$+ (b_0 + b_1 T + b_2 T^2 + b_3 T^3 + b_4 T^4) S$$

$$+ (c_0 + c_1 T + c_2 T^2) S^{3/2}$$

$$+ d_0 S^2 \tag{3.9}$$

### 3.2.2 Kinetic and Potential Energies

The evolution and transfer of energy in the wake is examined in the form of turbulent–kinetic, mean–kinetic, and potential energies defined as,

$$tke = \rho k, \quad ke = \frac{1}{2} \rho U^2, \quad pe = -\frac{1}{2} \frac{g}{\partial \rho_0 / \partial z} (\rho - \rho_0)^2 \tag{3.10}$$

$$TKE = \iint_S tke \, dS, \quad KE = \iint_S ke \, dS, \quad PE = \iint_S pe \, dS \tag{3.11}$$

The per–unit–volume energies $tke$, $ke$, and $pe$ can be integrated over an axial slice of area $S$ in the wake to find per–unit–length energy $TKE$, $KE$, and $PE$ as a function of downstream distance. The potential energy per–unit–volume $pe$ follows the formulation of Holliday and McIntyre [18].

### 3.2.3 Actuator–Line Model

The AL model works by projecting a line of force $f_p$ in the place of each propeller blade,

$$f_p(r) = \frac{F_p}{\varepsilon^{3/2} \pi^{3/2}} \exp \left[-\left(\frac{r}{\varepsilon}\right)^2\right] \tag{3.12}$$

where $F_p$ is the actuator element force composed of lift $L_p$ and drag $D_p$. The distance between CFD cell center and actuator point is $r$, and $\varepsilon$ controls the Gaussian width. Troldborg [36]
Chapter 3. Validation and Scaling of the Stratified Near Wake of a Self–Propelled Body

60

recommends $\varepsilon = 2\Delta x$ where $\Delta x$ is the grid spacing at the actuator position, but further considerations can be made [25]. For this study $\varepsilon = 4\Delta x$ to avoid numerical instability.

Lift and drag are computed from a lookup table of lift and drag coefficients $C_{\ell}$ and $C_d$ as functions of local flow angle of attack $\alpha$,

$$L = \frac{1}{2} C_{\ell}(\alpha) \rho U_{rel}^2 cw, \quad D = \frac{1}{2} C_d(\alpha) \rho U_{rel}^2 cw$$

where $\rho$ is the density, $U_{rel}$ is the local flow speed, $c$ is the chord and $w$ is the width of the actuator section. The relationship between $C_{\ell}$ and $C_d$ with $\alpha$ must be predetermined from experiment, simulation, or theory for each airfoil section. Figure 3.1 shows the projected body force on the AL propeller plane non–dimensionalized by $R_p rps^2$, where $R_p$ is the propeller radius and rps is the propeller rotations per second.

![Figure 3.1: Non–dimensional body, instantaneous force on mesh slice at propeller plane $f_p/(R_p rps^2)$](image)

3.2.4 Iowa Body

The Iowa Body, described in the experiment of Hyun and Patel [20], is shown in Figure 3.2. This axisymmetric geometry is representative of a typical marine vehicle without appendages. Features of this geometry are listed in Table 3.1.

![Figure 3.2: Iowa Body geometry](image)

The Iowa Body propeller is defined by 36 discrete sections to account for variations in radial propeller–blade geometry. Sectional lift $C_{\ell}$ is computed using the analytic expression of Brockett [4] for the NACA 66–Modified Airfoil,

$$C_{\ell} = 2\pi (1 - 0.83 \tau)(\alpha + 2.05f)$$

(3.14)
3.2. Approach

Table 3.1: Iowa Body geometry.

<table>
<thead>
<tr>
<th>Feature</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Body length ( l )</td>
<td>151.61 cm</td>
</tr>
<tr>
<td>Body diameter ( d )</td>
<td>13.91 cm</td>
</tr>
<tr>
<td>Hub radius ( r_h )</td>
<td>1.11 cm</td>
</tr>
<tr>
<td>Hub location</td>
<td>( 0.9688 &lt; x/l &lt; 0.9832 )</td>
</tr>
<tr>
<td>Propeller diameter ( d_p )</td>
<td>10.16 cm</td>
</tr>
<tr>
<td>Propeller location</td>
<td>( x/l = 0.9755 )</td>
</tr>
<tr>
<td>Number of blades</td>
<td>3</td>
</tr>
<tr>
<td>Propeller airfoil</td>
<td>NACA 66–Modified</td>
</tr>
</tbody>
</table>

where \( \alpha \) is the local flow angle of attack, \( \tau \) is the maximum thickness ratio, and \( f \) is the maximum camber ratio. Sectional drag \( C_d \) is imposed by combining viscous and induced drag at each airfoil section,

\[
C_d = C_{d0} + \frac{C_{e}^{2}}{\pi e AR} \tag{3.15}
\]

where \( C_{d0} \) is the viscous drag, \( e \) is the efficiency factor, and \( AR \) is the aspect ratio.

For the given operating conditions the local flow angle of attack typically remains below 1° and never exceeds 2.6°, so these analytic expressions do not need to account for stall. Pitch, chord, thickness, and camber distributions for the Iowa Body propeller blade are tabulated in Table 3.2. The Iowa Body propeller has zero rake and zero skew.

Table 3.2: Propeller blade radial geometry including chord, thickness, camber, pitch, and twist.

<table>
<thead>
<tr>
<th>Section</th>
<th>( r/R_p )</th>
<th>( c/D_p )</th>
<th>( t/D_p )</th>
<th>( CAM/D_p )</th>
<th>Pitch/( D_p )</th>
<th>Twist [°]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.219</td>
<td>0.3283</td>
<td>0.0320</td>
<td>0.00168</td>
<td>0.4139</td>
<td>31.0</td>
</tr>
<tr>
<td>5</td>
<td>0.325</td>
<td>0.3741</td>
<td>0.0271</td>
<td>0.00350</td>
<td>0.5544</td>
<td>28.5</td>
</tr>
<tr>
<td>10</td>
<td>0.450</td>
<td>0.4232</td>
<td>0.0218</td>
<td>0.00684</td>
<td>0.6826</td>
<td>25.8</td>
</tr>
<tr>
<td>15</td>
<td>0.575</td>
<td>0.4571</td>
<td>0.0169</td>
<td>0.00760</td>
<td>0.7724</td>
<td>23.2</td>
</tr>
<tr>
<td>20</td>
<td>0.700</td>
<td>0.4622</td>
<td>0.0125</td>
<td>0.00608</td>
<td>0.8120</td>
<td>20.3</td>
</tr>
<tr>
<td>25</td>
<td>0.825</td>
<td>0.4225</td>
<td>0.0083</td>
<td>0.00320</td>
<td>0.8051</td>
<td>17.3</td>
</tr>
<tr>
<td>30</td>
<td>0.950</td>
<td>0.2750</td>
<td>0.0047</td>
<td>0.00059</td>
<td>0.7470</td>
<td>14.1</td>
</tr>
<tr>
<td>35</td>
<td>1.000</td>
<td>0.0000</td>
<td>0.0033</td>
<td>0.00000</td>
<td>0.7110</td>
<td>12.8</td>
</tr>
</tbody>
</table>
3.2.5 Mesh

Mesh Generation

The $30 \times 10^6$ cell mesh shown in Figure 3.3 was generated using the software cfMesh [2]. The mesh was designed with nested refinement to focus cells near the body, the propeller region, and in the wake. Mesh design and quality features are listed in Table 3.3.

![Mesh with nested refinement](image1)

![Refined propeller region](image2)

![Boundary layer cells](image3)

Figure 3.3: Mesh generated with cfMesh [2].

Grid Study

A grid study was conducted for the case of the towed Iowa Body at the experimental-model scale. The observed order of accuracy is computed as $\ln \left( \frac{(f_3 - f_2)}{(f_2 - f_1)} \right) / \ln(r)$, where $r$ is the mesh refinement factor, $f_3$ is the solution on the coarsest mesh, $f_2$ is the solution on the medium-coarseness mesh and $f_1$ is the solution on the finest mesh. Results in Table 3.4 show that pressure drag converges with greater than first-order accuracy. Pressure drag is non-dimensionalized by the freestream density $\rho_0$, ship speed $U_0$, and ship wetted surface area $S_w$. The estimated discretization error in the computation of pressure drag is less than 6%. Viscous drag was not considered due to complexities in the systematic refinement of the boundary-layer cells.
Table 3.3: Mesh design and quality features.

<table>
<thead>
<tr>
<th>Mesh feature</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Boundary layer cells</td>
<td>&gt; 20</td>
</tr>
<tr>
<td>Maximum mesh spacing near wall at $Re_L = 310 \times 10^6$</td>
<td>$y^+ &lt; 100$</td>
</tr>
<tr>
<td>Cells per propeller diameter in propeller/wake regions</td>
<td>100</td>
</tr>
<tr>
<td>Wake region extends to</td>
<td>$x/L = 1.5$</td>
</tr>
<tr>
<td>Total number of cells</td>
<td>$N \approx 30 \times 10^6$</td>
</tr>
<tr>
<td>Maximum aspect ratio</td>
<td>AR $&lt; 170$</td>
</tr>
<tr>
<td>Maximum non-orthogonality</td>
<td>$&lt; 45$</td>
</tr>
<tr>
<td>Maximum skewness</td>
<td>$&lt; 0.8$</td>
</tr>
</tbody>
</table>

Table 3.4: Grid study.

<table>
<thead>
<tr>
<th>Mesh spacing $h$</th>
<th>Fine</th>
<th>Medium–Fine</th>
<th>Medium</th>
<th>Coarse</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$1$</td>
<td>$\sqrt{2}$</td>
<td>$2$</td>
<td>$2\sqrt{2}$</td>
</tr>
<tr>
<td>$(\text{pressure drag})/\left(\frac{1}{2}p_0U_0^2S_w\right)$</td>
<td>$0.0003145$</td>
<td>$0.0003344$</td>
<td>$0.0003656$</td>
<td>$0.0004105$</td>
</tr>
<tr>
<td>Observed order of accuracy</td>
<td>$1.3$</td>
<td>$1.1$</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

3.2.6 Flow Field Analysis

This study examines primary flow variables including: axial, radial, and azimuthal velocities $U_x$, $U_r$, and $U_\theta$; change in temperature $\Delta T$; and $\text{tke}$, $\text{ke}$, and $\text{pe}$. Additionally, the following secondary flow variables are examined: pressure, skin friction, and total drag coefficients $C_p$, $C_f$, and $C_D$; vorticity and $Q$–criterion visualization; and PE, KE, and TKE. Data is extracted in axial planes between $x/L = 0.782$ and $x/L = 1.45$ where $x$ is the axial distance from the start of the body and $L$ is the body length.

3.2.7 Flow Coefficients and Case Studies

Table 3.5 lists the most important flow coefficients for propeller–driven flow including Reynolds number $Re_L$, advance ratio $J$, thrust coefficient $C_T$, and torque coefficient $C_Q$.

$$Re_L = \frac{U_0L}{\nu}, \quad J = \frac{U_0}{nD_p}, \quad C_T = \frac{F_T}{\rho_0n^2D_p^4}, \quad C_Q = \frac{T_Q}{\rho_0n^2D_p^5}$$

For these expressions, $\nu$ is kinematic viscosity, $D_p$ is the diameter of the propeller, $n$ is the propeller speed in revolutions per second, $F_T$ is the thrust, and $T_Q$ is the torque. With an increased Reynolds number the boundary layer of the Iowa Body becomes thinner, thereby increasing the flow velocity experienced by outer regions of the propeller. This relatively
higher velocity results in a higher local angle of attack $\alpha$, decreasing the required thrust coefficient. The required thrust coefficient is further decreased by the reduced skin–friction coefficient due to the thinned boundary layer. Because of these factors $J$, $C_T$, and $C_Q$ vary with $Re_L$.

Table 3.5: Flow coefficients.

<table>
<thead>
<tr>
<th>$Re_L$</th>
<th>$J$</th>
<th>$C_T$</th>
<th>$C_Q$</th>
<th>$C_T/C_D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$1.6 \times 10^6$</td>
<td>0.812</td>
<td>0.0843</td>
<td>0.0161</td>
<td>1.31</td>
</tr>
<tr>
<td>$3.1 \times 10^8$</td>
<td>0.855</td>
<td>0.0516</td>
<td>0.0105</td>
<td>1.09</td>
</tr>
</tbody>
</table>

The individual cases covered in this study are listed in Table 3.6. Experimental–model–scale simulations at $Re_L = 1.6 \times 10^6$ were conducted and compared to experimental data, while simulations at $Re_L = 310 \times 10^6$ were run to consider the effects of increasing Reynolds number and introducing stratification. The internal Froude number $Fr$ provides a measure of the density stratification, where $Fr = 1$ means zero stratification and a small $Fr$ means high levels of stratification. The stratified case in the present study considers a Froude number of $Fr = 350$, where background temperature is linearly stratified and salinity is held constant.

$$Fr = \frac{1}{N} \frac{U_0}{D}, \quad \text{where} \quad N = \frac{1}{2\pi} \sqrt{-g \frac{\partial \rho}{\rho_0 \partial z}}$$

(3.16)

In these expressions $N$ is the Brunt Väisälä frequency, $g$ is acceleration due to gravity, and $z$ is the vertical coordinate. The impact of buoyancy on the near–wake fluid dynamics is often small and can be explained by the Richardson number $Ri$, which is the ratio of buoyancy to flow gradient terms [24].

$$Ri = \frac{g}{T_0} \frac{dT/dz}{(dU/dz)^2}$$

(3.17)

$T_0$ is a reference temperature, $T$ is the mean temperature, and $U$ is the mean velocity. In this case, $Ri \approx 2.54 \times 10^{-3}$ which means the near–wake inertial forces of the propeller dominate the buoyancy forces. As the local velocity $U_x$ decays in the wake, $Ri$ increases and buoyancy forces become important.

### 3.2.8 Characterizing Vortex Path

Characterization of the vortex path is not a straightforward task. The paths of root and tip vortices are considered. These vortices are visualized by taking iso–surfaces of the $Q$–criterion. The geometric vortex locations are found through the process illustrated in Figure 3.4. First the iso–surface of the total vortex structure is sliced at a given downstream position. Next the root or tip vortex of interest are selected by hand. This data is then be
3.2. Approach

Table 3.6: Case studies.

<table>
<thead>
<tr>
<th>Case</th>
<th>( \text{Re}_L )</th>
<th>Fr</th>
<th>Self-propelled?</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>( 1.6 \times 10^6 )</td>
<td>( \infty )</td>
<td>no</td>
</tr>
<tr>
<td>2</td>
<td>( 1.6 \times 10^6 )</td>
<td>( \infty )</td>
<td>yes</td>
</tr>
<tr>
<td>3</td>
<td>( 3.1 \times 10^8 )</td>
<td>( \infty )</td>
<td>no</td>
</tr>
<tr>
<td>4</td>
<td>( 3.1 \times 10^8 )</td>
<td>( \infty )</td>
<td>yes</td>
</tr>
<tr>
<td>5</td>
<td>( 3.1 \times 10^8 )</td>
<td>350</td>
<td>no</td>
</tr>
<tr>
<td>6</td>
<td>( 3.1 \times 10^8 )</td>
<td>350</td>
<td>yes</td>
</tr>
</tbody>
</table>

used to compute the vortex center at that downstream position \( x \) in terms of radial position \( r \) and azimuthal angle \( \theta \). After taking enough slices of axial data, the vortex paths are reconstructed to compute \( x - r \) and \( x - \theta \) relationships.

![Figure 3.4](image)

(a) 1. Slice wake  
(b) 2. Select vortex  
(c) 3. Compute center

Figure 3.4: Process of extracting downstream positional data of root- and tip-vortices.

3.2.9 Coupling Among Vortices

Coupling between propeller root- and tip-vortices is found by examining their spatial evolution in the wake. By comparing their \( x - r \) curves, a correlation coefficient is computed as,

\[
\rho_{\text{tip},\text{root}} = \frac{\text{Cov}(r_{\text{tip}}, r_{\text{root}})}{\text{Cov}(r_{\text{tip}}, r_{\text{tip}}) \text{Cov}(r_{\text{root}}, r_{\text{root}})}
\]

(3.18)

where \( \text{Cov} \) is the covariance. This coefficient describes the degree of correlation between these two vortex paths where a coefficient value of \( \rho_{\text{tip},\text{root}} = 1 \) implies perfect correlation and \( \rho_{\text{tip},\text{root}} = -1 \) implies perfect anti-correlation. For perfectly correlated functions, a linear equation can describe the relationship between the two; as one increases the other increases proportionally. For anti-correlation, as one function increases the other decreases.
3.3 Comparison to Experiment

3.3.1 Pressure Coefficient

Figure 3.5 compares the unpropelled “towed” hull pressure at \( Re_L = 1.6 \times 10^6 \) and \( Re_L = 310 \times 10^6 \) with the experimental data of Hyun and Patel [20]. The pressure coefficient is defined as \( C_p = 2(p - p_0)/\rho_0 U_0^2 \), and \( x' \) refers to the axial coordinate referenced to zero at the end of the body. The results at the experimental–model scale with \( Re_L = 1.6 \times 10^6 \) indicate strong agreement giving confidence that the flow conditions and overall geometry are properly defined.

![Figure 3.5: Pressure coefficient over unpropelled Iowa Body compared to the experimental data of Hyun and Patel [20].](image)

3.3.2 Vehicle Boundary Layer

Figure 3.6 compares the simulated boundary layer at both low and high Reynolds numbers with the experimental measurements of Hyun and Patel [20]. The axial velocity \( U_x \) which is non–dimensionalized by the freestream velocity \( U_0 \) is plotted against distance from the hull wall \( r - r_0 \) non–dimensionalized by the hull radius \( R_0 \). Results at the experimental–model scale with \( Re_L = 1.6 \times 10^6 \) compare favorably with the measured data yet over–predict the boundary layer thickness. Additional simulations showed that this discrepancy was the result of the experimental model being placed within the wind tunnel contraction, whereas in the simulations, the far–field boundaries were located further away and did not match the geometry of the tunnel contraction and exit. The wind–tunnel contraction and exit played a role in modifying the flow field, which further modified the hull boundary layer.
3.3. Comparison to Experiment

Figure 3.6: Boundary layer profiles on the body compared to the experimental data of Hyun and Patel [20].

3.3.3 Phase–Averaged Contours

The flow field at the axial positions $x/L = 1.01, 1.04, 1.10, \text{ and } 1.30$ is compared with the phase–averaged experimental measurements of Hyun and Patel [20] in Figures 3.7, 3.8, and 3.9 for axial velocity, axial vorticity, and turbulent kinetic energy, respectively. The axial velocity, non–dimensionalized by $U_0$ resembles the experiment qualitatively, but loses flow details beyond $x/L = 1.10$. Similar observations are drawn for the axial vorticity, non–dimensionalized by $-U_0/R_p$. The negative value is used for consistency because the experimental propeller rotated clockwise (looking upstream), whereas the simulated propeller rotates counter–clockwise. The three root vortices rapidly dissipate in the experiment, while they are still discernible in the simulation at $x/L = 1.10$. Finally, the turbulence quantity $10^3k/U_0^2$ shows the poorest comparison. Experimental results show the distinct regions of each propeller blade, while $k$ of the simulation homogenizes more rapidly. This latter point is due to the lack of shear production in the propeller blade boundary layers, which is not resolved with the AL model.

It is possible to improve simulation fidelity in two major ways. First, a fully geometrically–resolved rotating propeller could be implemented to capture propeller blade boundary layer effects. Second, a LES or hybrid RANS/LES approach could be implemented to better model turbulence. The downside of these fidelity improvements is a major increase in computational cost. At present time, these methods of simulation are not feasible for large numbers of simulations where many vehicle concepts are considered.
Chapter 3. Validation and Scaling of the Stratified Near Wake of a Self-Propelled Body

Figure 3.7: Experimental–model–scale non–dimensional axial velocity $U_x/U_0$ at various axial locations compared to the experimental measurements of Hyun and Patel [20].

Figure 3.8: Experimental–model–scale non–dimensional axial vorticity $-\omega_x R_p/U_0$ at various axial locations compared to the experimental measurements of Hyun and Patel [20].
3.3. Comparison to Experiment

(a) AL model. 
(b) Experiment [20].

Figure 3.9: Experimental–model–scale non–dimensional turbulent kinetic energy $10^3k/U_0^2$ at various axial locations compared to the experimental measurements of Hyun and Patel [20].

3.3.4 Circumferentially–Averaged Wake Profiles

Finally, circumferentially–averaged profiles of the velocity field are compared with the experimental measurements of Hyun and Patel [20]. Figure 3.10 compares the radial distribution of axial velocity, non–dimensionalized by $U_0$, at both low and high Reynolds numbers at the axial locations $x/L = 1.01, 1.10,$ and $1.30$. In the unpropelled “towed” simulation, the AL model is disabled such that the body wake exhibits pure drag. The AL model simulation represents the case where a rotating propeller provides nearly equal–and–opposite thrust to offset drag thereby resulting in a net–zero–momentum wake. The region of positive momentum in the wake can be recognized by the “lobe” of axial velocity where $U_x/U_0 > 1$ at $r/R_p \approx 0.6$, while the negative momentum regions exist where $U_x/U_0 < 1$. At $x/L = 1.01$, the results for $Re_L = 1.6 \times 10^6$ show strong agreement to experiment. The towed profile over–predicts the wake shear layer, but this discrepancy is likely due to wind–tunnel test–section wall effects that were neglected in the simulation. Further downstream, however, the results vary more noticeably as the simulation shows increased centerline velocity decay. This excess dissipation is thought to be a result of the turbulence model.

Figure 3.10 also shows the circumferentially–averaged azimuthal “swirl” velocity profile. Simulation results at the experimental–model scale with $Re_L = 1.6 \times 10^6$ show qualitative agreement but exhibit variations quantitatively. At $x/L = 1.01$, the simulated swirl profile omits a distinctive spike seen in the experiment. Across all downstream positions simulated swirl is higher in magnitude and the locations of maximum swirl vary from those of the experiment.
Figure 3.10: Circumferentially-averaged axial and swirl velocity profiles at $x/L = 1.01, 1.10, 1.30$. 
3.4 Results

3.4.1 Buckingham Π Dimensional Analysis

Buckingham’s Π method, originally proven by Buckingham [6] in 1914, can be reviewed in the works of Evans [14] applied to the heat equation, and Curtis et al. [13], whose proof extends the method of Birkhoff [3]. Here, Buckingham’s Π method is applied to the near-field formulation of a stratified wake profile following the procedure enumerated in Cengel and Cimbala [7]. The independent variable is selected as $\Delta \rho$. The dependent variables are selected as $U_0$, $\frac{\partial U}{\partial z}$, $L$, $\mu$, $g$, $\rho_0$, and $\frac{\partial \rho}{\partial z}$. In total there are 8 parameters. Because the parameters inhabit three dimensions: mass, length, and time, the reduction is chosen to be 3. So, there will be $8 - 3 = 5$ Πs. Given the reduction, the 3 repeating parameters are chosen to be $U_0$, $L$, and $\rho_0$, resulting in the following 4 Πs.

$$
\Pi_1 = \frac{\Delta \rho}{\rho_0} \\
\Pi_2 = \frac{\partial U / \partial z}{U_0 / L} \\
\Pi_3 = \frac{U_0 L}{\nu} \\
\Pi_4 = \frac{U_0}{\sqrt{gL}} \\
\Pi_5 = \frac{\partial \rho}{\partial z} \frac{L}{\rho_0}
$$

(3.19)

So, $\Pi_1 = \Delta \rho / \rho = f(\Pi_2, \Pi_3, \Pi_4, \Pi_5)$. It is clear that the third Π is the Reynolds number $\Pi_3 = Re$, or the ratio of inertial to viscous forces. The other Πs are manipulated through standard algebraic operations to yield similarly familiar results. Dividing $\Pi_5$ by $\Pi_2^2$ and $\Pi_4^2$ reveals the Richardson number, which is the ratio of buoyancy to flow shear.

$$
\Pi_5 \Pi_2^2 \Pi_4^2 = \frac{1}{\rho_0} \frac{\partial \rho}{\partial z} \frac{\partial U / \partial z}{\rho_0} = Ri
$$

(3.20)

Manipulating $\Pi_4$ and $\Pi_5$ reveals the internal Froude number, which is the ratio of inertial to gravitational forces, provides a measure of the density stratification.

$$
\frac{1}{2\pi} \frac{\Pi_4}{\sqrt{\Pi_5}} = \frac{1}{2\pi} \frac{U_0}{\sqrt{g \frac{\partial \rho}{\partial z}}} = Fr
$$

(3.21)

Noticing that $\Pi_3$ is the Reynolds number the final relationship is,

$$
\frac{\Delta \rho}{\rho_0} = f(Re, Ri, Fr)
$$

(3.22)
This dimensional analysis reveals that the formation of the stratified wake density profile is a function of Reynolds number, Richardson number, and internal Froude number. Note that because $\Delta T$ is directly proportional to $\Delta \rho$ through the equation of state, $\Delta T \sim \Delta \rho$, a similar conclusion can be drawn for the $\Delta T$ wake profile.

$$\frac{\Delta T}{\Delta T_{R_p}} = f(Re, Ri, Fr)$$

(3.23)

Dimensional analysis reveals the functional relationship between these flow fields and other non-dimensional parameters, however, it does not provide the form of the equation. To understand the effect of these parameters, additional analysis is required.

### 3.4.2 Reynolds Number Effects

Several Reynolds number effects are recognized when increasing $Re_L = 1.6 \times 10^6$ to $Re_L = 310 \times 10^6$. First, the hull boundary-layer of Figure 3.6 should be considered. Results indicate that the boundary-layer thickness decreases at higher Re, a phenomenon predicted from theory. The decreased boundary-layer thickness most noticeably effects the propeller blade inflow conditions, which decreases the outboard angle of attack and changes the radial loading distribution.

Next, the pressure and skin-friction coefficients can be examined in Figures 3.5 and 3.11 respectively. Differences in $C_p$ are small, but deviations do appear at points of strong curvature in the hull geometry. This is a direct result of the thinner boundary layer. The skin-friction coefficient is defined by $C_f = 2\tau_w/\rho U_0^2$, where $\tau_w$ is the wall shear stress on the body. Overall $C_f$ follows the same shape but decreases by roughly a factor of two due to the increased Reynolds number. These $C_f$ distributions are compared to White’s empirical equation for the $C_f$ over a flat plate [10],

$$C_f = 0.455 \ln [0.06 \text{Re}_x]^2$$

(3.24)

where $\text{Re}_x$ is the Reynolds number with respect to axial position. The computed axisymmetric Iowa Body $C_f$ follows similar changes in magnitude to that of White’s flat plate empirical formula. Variations occur at regions of high curvature, but less so at $Re_L = 310 \times 10^6$ showing that form effects are small.

Reynolds number effects on the wake can be seen in the circumferentially-averaged velocity profiles of Figure 3.10. Axial velocity of the unpropelled wakes is shown to be weaker, while that of the propelled case is stronger when compared to the freestream $U_0$. This result shows that the differences between the propelled and unpropelled wake profiles are lessened at higher Reynolds numbers. Centerline values are closer in magnitude. Comparison of swirl velocity $U_\theta/U_0$ shows that the swirl magnitude is decreased with an increased Reynolds number, yet similar structure remains.
3.4. Results

Finally the effects of Reynolds number are observed visually in Figure 3.12, which extends to $x/L = 1.3$. Isosurfaces of the second invariant of the velocity gradient tensor $Q$ offers a method of vortex identification. Here $Q$ is non-dimensionally set to $(L/U_0)^2 Q = 1.7$. The visualizations show the three helical propeller-tip vortices spiraling around the diameter of the wake in addition to the propeller-root vortices near the body axis. These vortices break down more rapidly at $Re_L = 1.6 \times 10^6$ than at $Re_L = 310 \times 10^6$. It should be noted that Chase and Carrica [8] has shown the RANS turbulence model to be overly dissipative in the wake, so future studies are required to examine higher fidelity turbulence modeling.

Figure 3.12: $Q$-criterion flow visualization with $(L/U_0)^2 Q = 16.9$ extending to $x/L = 1.3$.

3.4.3 Characterizing Vortex Paths

Using the method described in Section 3.2.8, the geometric centers of the root and tip vortices for the $Re_L = 310 \times 10^6$ case. Figure 3.13 shows $x - r$ and $x - \theta$ curves for these vortical paths. The tip vortex decreases nearly monotonically until it is no longer detectable, which
is thought to be the point of vortex breakdown. Some noise is present in the curve likely as a result of imperfections in describing the vortex shape on a Cartesian mesh as it moves linearly in the azimuthal direction \( \theta \). The root vortex radial position \( r_{\text{root}} \) increases up until a maximum value shortly before the tip–vortex breakdown and then decreases until \( x/L = 1.4 \). Beyond \( x/L = 1.5 \) distinct root vortices are no longer distinguishable and only the rotating hub–vortex–core is present, which steadily increases.

![Radial and azimuthal position of root and tip vortices for \( Re_L = 310 \times 10^6 \).](image)

Figure 3.13: Radial and azimuthal position of root and tip vortices for \( Re_L = 310 \times 10^6 \).

### 3.4.4 Coupling Among Vortices

Equation 3.18 is used to compute correlation coefficients between root- and tip- vortical paths as listed in Table 3.7. Prior to the peak of \( r_{\text{root}} \) at \( x/L \approx 1.13 \), the two paths show anti–correlation with \( \rho_{\text{tip},r_{\text{root}}} = -0.889 \). After this maximum, however, the two paths are positively correlated with \( \rho_{\text{tip},r_{\text{root}}} = 0.877 \). This level of correlation suggests that shortly beyond \( x/L = 1.13 \), the root- and tip- vortices are interacting with one another, which may initiate breakdown at \( x/L = 1.24 \).

<table>
<thead>
<tr>
<th>Region</th>
<th>( \rho_{\text{tip},r_{\text{root}}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( 1.00 &lt; x/L &lt; 1.13 )</td>
<td>-0.889</td>
</tr>
<tr>
<td>( 1.13 &lt; x/L &lt; 1.24 )</td>
<td>0.877</td>
</tr>
</tbody>
</table>

Table 3.7: Correlation between root and tip vortical paths
3.4. Results

3.4.5 Evolution of Kinetic and Potential Energies

The axial evolution of cross–plane–integrated turbulent–kinetic and mean–kinetic energies is shown in Figure 3.14, while potential energy $PE$ is shown in Figure 3.15a. Moving downstream through the wake, each component of the total–mean–kinetic energy $KE$ decays, but the radial and axial components $KE_r$ and $KE_x$ decays most rapidly. The azimuthal swirl $KE_\theta$ notably decays the slowest and is shown to persist. The turbulent–kinetic energy $TKE$ increases as the wake circulations expand and vortices break down, and $PE$ steadily increases beyond $x/L = 1.1$. For $x'/D_p \geq 4$, the energies follow power–law slopes suggesting that the wake is transitioning from the intermediate region to the far–wake.

![Figure 3.14: Axial development of turbulent–kinetic, mean–kinetic, and potential energies.](image)

Power–law curve fits were established following the formula $\mathcal{E} = a(x/L)^b$, where $\mathcal{E}$ is the particular non–dimensional $KE$ or $PE$ energy term, and $a$ and $b$ are the related constants listed in Table 3.8. Note that $L/D_p = 14.92$ for the Iowa Body so the expression $x'/D_p = 14.92(x/L - 1)$ converts $x/L$ to $x'/D_p$, which is zero–referenced to the end of the body. By
Chapter 3. Validation and Scaling of the Stratified Near Wake of a Self–Propelled Body

76

(a) Maximum magnitudes. (b) Radial distances to relevant wake features.

Figure 3.15: Axial development of relevant wake features.

$x'/D_p = 30$, $KE_r$ is predicted to be of the same order as $PE$ and so is $KE_x$ by $x'/D_p = 45$. However, $KE_\theta$ remains much larger than $PE$, suggesting the importance of swirl in the far–wake evolution.

Table 3.8: Power law with $\mathcal{E} = a(x/L)^b$.

<table>
<thead>
<tr>
<th>$\mathcal{E}$</th>
<th>$a$</th>
<th>$b$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$KE_r \times 10^8/\rho_0L^2U_0^2$</td>
<td>1100</td>
<td>-10</td>
</tr>
<tr>
<td>$KE_\theta \times 10^8/\rho_0L^2U_0^2$</td>
<td>1400</td>
<td>-1.5</td>
</tr>
<tr>
<td>$KE_x \times 10^8/\rho_0L^2U_0^2$</td>
<td>2650</td>
<td>-6.8</td>
</tr>
<tr>
<td>$PE \times 10^8/\rho_0L^2U_0^2$</td>
<td>0.516</td>
<td>+1.9</td>
</tr>
</tbody>
</table>

Several points of interest in the wake are examined including the point of: maximum potential energy, $\max(|\Delta T/\Delta T_{R_p}|)$, maximum swirl $\max(U_\theta/U_0)$, maximum positive axial momentum $\max(U_x/U_0)$, and the wake outer boundary defined where the swirl velocity reaches zero $U_\theta/U_0 = 0$. Magnitudes are shown in Figure 3.15a, while radial distances to these points are shown in Figure 3.15b. These values help to explain the spatial evolution of the wake. Even though the radial distance of maximum axial momentum remains relatively constant, the potential energy increases due to the expansion of the wake diameter.

3.4.6 Evolution and Structure of the Mixed Patch

A mixed region of temperature in the linearly stratified environment emerges downstream of the propeller due to the induced swirl of the wake. This transportation of the background
3.4. Results

temperature field can be visualized through temperature deviation $\Delta T$ non-dimensionalized by the stratified background temperature deviation over the height of the propeller radius, $\Delta T_{R_p}$. Figure 3.16 shows the progression of $\Delta T/\Delta T_{R_p}$ at various axial cross-planes downstream of the hull. Unsteadiness dominates close to the propeller, but a steady profile emerges by $x/L = 1.45$. Figure 3.17 helps to understand the emergence of the mixed patch by showing an iso-surface of the temperature at the background centerline value. Counter-clockwise swirl in the wake transports the temperature field into the quasi-steady shape shown.

![Figure 3.16: Instantaneous and time-averaged temperature deviation $\Delta T/\Delta T_{R_p}$ cross-plane surfaces at $x/L = 1.01, 1.10, 1.30, 1.45.$](image)

(a) Instantaneous. (b) Time-averaged.

Figure 3.16: Instantaneous and time-averaged temperature deviation $\Delta T/\Delta T_{R_p}$ cross-plane surfaces at $x/L = 1.01, 1.10, 1.30, 1.45.$

![Figure 3.17: Instantaneous iso-surface of the temperature at the background centerline value for $0.98 \leq x/L \leq 1.45.$](image)

Figure 3.17: Instantaneous iso-surface of the temperature at the background centerline value for $0.98 \leq x/L \leq 1.45.$

The mixed patch field $\Delta T/\Delta T_{R_p}$ is presented at $x/L = 1.45$ in Figure 3.18a. Colder fluid is entrained and transported to the top of the wake, while warmer fluid is pulled to the bottom. Figure 3.18b shows a theoretical “perfectly-mixed” mixed patch, in which temperature is
constant across the entire wake disk. Comparison between these two figures shows that the simulated temperature deviation is similar in magnitude and shape to that of a theoretical perfectly–mixed profile. However, the actual entrainment and distribution of the passive scalar $T$ leads to an asymmetric profile in $\Delta T/\Delta T_{R_p}$.

(a) Simulation. 
(b) Perfectly mixed.

Figure 3.18: Instantaneous, on–dimensional temperature deviation $\Delta T/\Delta T_{R_p}$ at $x/L = 1.45$ compared to a theoretical, perfectly–mixed profile.

3.4.7 Internal Froude Number Independence

The overall low Richardson number of the flow shows that flow–gradient inertial forces are large compared to buoyancy forces. Consequently the mixed–patch profile of $\Delta T/\Delta T_{R_p}$ is independent of stratification intensity so long as it is properly non–dimensionalized by the level of intensity. For this study the temperature deviation is non–dimensionalized by $\Delta T_{R_p}$, that is, the temperature change over the depth of one propeller diameter. Figure 3.19 illustrates this effect in the mixed patch of the vehicle. The temperature deviation $\Delta T/\Delta T_{R_p}$ is plotted versus depth at half of a body length downstream. Internal Froude number is varied by increasing or decreasing the linear background temperature gradient. For each Froude number, the curves collapse on one another when non–dimensionalized by their respective stratification intensities, demonstrating the dominance of flow gradient forces over buoyancy forces and independence to Froude number. Although buoyancy forces are insignificant in the near field, they may become significant in the far field.
3.4. RESULTS

Figure 3.19: Center–plane depth–wise deviation to temperature $\Delta T/\Delta T_{R_p}$ for various stratification intensities at $x/L = 1.5$.

3.4.8 Analytical Far–Field Profiles

By $x/L = 1.45$, $U_x$, $U_\theta$ and $k$ have lost the distinct signatures of each propeller blade and are almost entirely mixed in the azimuthal direction. Thus, they may be described solely by the radial coordinate $r$. Figure 3.20 shows simulated contour surfaces at $x/L = 1.45$. Figure 3.21 compares simulated radial profiles to analytical curve fits. Using the constants: $A_x = 58$, $B_x = 31.1$, $A_\theta = 110$, $B_\theta = 40$, $C = 16$, and $F = 10$, the profiles are fit to the following analytical equations,

$$U_d = U_{d0} \left( 1 - A_x \left( \frac{r}{D_w} \right)^2 \right) \exp \left[ -B_x \left( \frac{r}{D_w} \right)^2 \right]$$

$$U_\theta = U_{\theta,max} A_\theta \left( \frac{r}{D_w} \right)^2 \exp \left[ -B_\theta \left( \frac{r}{D_w} \right)^2 \right]$$

$$k = k_0 \exp \left[ -C \left( \frac{r}{D_w} \right)^2 \right] + k_b$$

$$\omega = \omega_0 \exp \left[ -F \left( \frac{r}{D_w} \right)^2 \right] + \omega_b$$

(3.25)

where $U_d = U_x - U_0$ is the velocity defect, $U_{d0}$ is the centerline velocity defect, $U_{\theta,max}$ is the maximum swirl velocity, $k_0$ is the centerline $k$ value, $\omega_0$ is the centerline $\omega$ value, and $D_w$ is the wake diameter. These profiles, along with the mixed patch field, may be used as the initial conditions in a far–field simulation.
Figure 3.20: Instantaneous contour surfaces for $Re_L = 310 \times 10^6$ at $x/L = 1.45$.

Figure 3.21: Comparison between analytical and simulated profiles for $Re_L = 310 \times 10^6$ at $x/L = 1.45$. 
3.4.9 Mixed–Patch Quasi–Analytical Profile

The $\Delta T/\Delta T_{R_p}$ profile is distinct from the fully–mixed axi–symmetric velocity and turbulence profiles at $x/L = 1.45$ due to its asymmetry in the azimuthal direction. A novel analytical method has been formulated to model the mixed–patch temperature profile as described in Appendix A and shown in Figure 3.22a. Figure 3.22b shows radial distributions of $\Delta T/\Delta T_{R_p}$ angles corresponding to the azimuthal direction where $90^\circ$ is in the positive $z$ direction, $0^\circ$ is in the positive $y$ direction, and $-90^\circ$ is in the negative $z$ direction. The mixed–patch profile is imitated by constructing a temperature profile using the Fermat Spiral mathematical construct. This analytical approach to describing the mixed–patch temperature profile shows some discrepancies with the simulated profile. First, discontinuities in the gradient at local maxima. Second, the temperature–deviation field does not decay properly at the wake boundary. This shortcoming motivates a better method for modeling the profile, namely through machine learning.

(a) Quasi–analytical model contours. (b) Comparison to simulated AL model results.

Figure 3.22: Quasi–analytical model of the mixed–patch temperature–deviation field compared with simulated results for various azimuthal directions.

3.5 Conclusions

A hierarchy of complexity exists in techniques used to simulate propelled flows. Due to the interaction of turbulence and vorticity, simplistic approaches are often insufficient, but increased algorithmic fidelity comes with increased computational expense. The AL model offers a compromise in which dominant flow physics of the near–wake are captured, while
implementation effort and expense are relatively small. Although the AL model is typically used in the study of wind turbines, this study presents a novel use in the study of a transient, self-propelled near-wake of the axisymmetric Iowa Body. The experiment of Hyun and Patel [20] is reproduced at \( Re_L = 1.6 \times 10^6 \), and Reynolds number effects are studied by comparing to the case of \( Re_L = 310 \times 10^6 \). As predicted from theory, the boundary-layer thickness and skin-friction coefficient decrease. With increased Reynolds number, the simulations also show that the magnitudes of the axial-defect and swirl velocities are reduced, and that the vortex breakdown is delayed. The flow field is shown to be independent of internal Froude number. The primary focus of the study, however, is on the impact of a rotating propeller on the near wake in a linearly stratified environment. The evolution of turbulent-kinetic, mean-kinetic, and potential energies show that while \( KE_x \) and \( KE_r \) decrease downstream in the wake, \( TKE, PE \), and wake diameter increase. Furthermore, \( KE_\theta \) persists, which suggests the importance of swirl in the far field. By \( x/L = 1.45 \), or \( x'/D_p = 6.71 \), the wake is almost entirely mixed allowing for the curve-fit of axisymmetric analytic profiles of \( U_x, U_\theta \), and \( k \) as functions of \( r \). The temperature-deviation field, however, is asymmetric about the centerline axis, yielding a unique distribution that has been imitated with a newly developed formulation.
Bibliography


Appendix A  Analytical Profile for Mixed–Patch Temperature Deviation

This model constructs the mixed–patch temperature profile with the aid of the Fermat Spiral mathematical construct. To start, the following expression gives the Fermat’s Spiral radial position \( r \) as a function of angular position \( \theta \).

\[
r = k\theta^{1/n}
\] (3.26)

Here, \( k \) is a scaling constant, and \( n \) determines how tightly the spiral is wrapped. An Archimedean spiral uses \( n = 2 \).

\[
r = k\sqrt{\theta}
\] (3.27)

Variations in \( n \) affect the compactness of the spiral.

In Cartesian coordinates, the expression for Fermat’s Spiral is,

\[
y = r \cos(\theta) = k\theta^{1/n} \cos(\theta)
\]
\[
z = r \sin(\theta) = k\theta^{1/n} \sin(\theta)
\] (3.28)

With an initial tangent angle \( \chi_0 \) and assigning \( \gamma = 1 \) for a counter–clockwise (CCW) spiral and \( \gamma = -1 \) for clockwise (CW) spiral the expression becomes,

\[
y = k\theta^{1/n} \cos(\gamma\theta + \chi_0)
\]
\[
z = k\theta^{1/n} \sin(\gamma\theta + \chi_0)
\] (3.29)

Figure A.1a shows variations in the initial angle \( \chi_0 \). Looking solely at the \( \chi_0 = 0^\circ \) and \( \chi_0 = 90^\circ \) curves, a shape similar to the mixed–patch temperature profile can be discerned. The shape is incomplete, however, because it grows unbounded in the radial direction.

In order to bound the profile shape to the radius of the wake \( R_w \), the equations are further modified by adding in a bounding term and relaxation term \( \eta \).

\[
\eta(\theta) = \begin{cases} 
1 - \theta/(2\pi), & \text{if } 0 < \theta < 2\pi \\
0, & \text{if } 2\pi \leq \theta
\end{cases}
\]
\[
y = \eta k^{1/n} \cos(\gamma\theta + \chi_0) + (1 - \eta)R_w
\]
\[
z = \eta k^{1/n} \sin(\gamma\theta + \chi_0) + (1 - \eta)R_w
\] (3.30)

With this modification, the spiral will never grow beyond \( r = R_w \). If \( \theta > 2\pi \) then \( r = R_w \). Adding two spirals with initial angles \( \chi_0 \) 180° apart, with one CW and the other CCW, the shape of the mixed patch can be discerned in Figure A.1b.

The final step is to consider the variation of the scalar temperature field within the mixed–patch profile. To best imitate this distribution, temperature is normalized by distance to nearest spiral curve. The upper region is assigned a negative value, while the bottom is positive. Finally, the entire plot is rotated by \( \approx 5^\circ \) CCW. The end result is shown in Figure 3.22a.
A. Analytical Profile for Mixed–Patch Temperature Deviation

(a) Variation of $\chi_0$.

(b) Expression modified to approach wake radius.

Figure A.1: Variations in Fermat Spiral parameters.
Chapter 4

Influence of Propulsion Type on the Stratified Near Wake of a Self–Propelled Body

(Work adapted from Jones and Paterson [15, 17].)

Abstract

To better understand the influence of swirl on the thermally–stratified near wake of a self–propelled axisymmetric vehicle, three propulsor schemes are considered: a single propeller, contra–rotating propellers (CRP), and a zero–swirl, uniform–velocity jet. The propellers are implemented using an Actuator–Line model in an unsteady Reynolds–Averaged Navier–Stokes simulation, where the Reynolds number is $Re_L = 310 \times 10^6$ based on the freestream velocity and body length. The authors have previously shown good comparison to experimental data with this approach. Visualization of vortical structures shows the helical paths of blade–tip vortices from the single propeller as well as the complicated vortical interaction between contra–rotating blades. Comparison of instantaneous and time–averaged fields shows that temporally stationary fields emerge by half of a body length downstream. Circumferentially–averaged axial velocity profiles show similarities between the single propeller and CRP in contrast to the jet configuration. Swirl velocity of the CRP, however, was attenuated in comparison to that of the single propeller case. Mixed–patch contour maps illustrate the unique temperature distribution of each configuration as a consequence of their respective swirl profiles. Finally, kinetic and potential energy is integrated along downstream axial planes to reveal key differences between the configurations. The CRP configuration creates less potential energy by reducing swirl that would otherwise persist in the near field of a single–propeller wake.
4.1 Introduction

Experiments show that propeller–driven wakes evolve from a complicated near wake with discernible propeller–blade features, to a far wake, in which these features have mixed together to form a nearly–axisymmetric field \([12, 13]\). Sirviente and Patel [34] show that the near–wake region transitions to the far wake in roughly twelve initial propulsor diameters, but the development of the far wake can be delayed by appendages on the body [29]. This transition is influenced by the Reynolds number, body geometry, and operation of the propulsor [8], which itself has a large impact on the ingested stern boundary layer and downstream turbulence [30, 31]. The swirling propeller induces helical vortices that are shed from the roots and tips of the individual blades. In the near wake, these vortices break down, which is a topic of extensive study [20]. Although experiments show the contribution of swirl [35], its role in the evolution from near to far wake is not well–characterized.

In a stratified wake, a mixed patch is formed by swirl from the propeller, turbulent mixing, and potential effects from the upstream body [7]. This mixed patch can further modify the far wake in the event of a mixed–patch collapse when buoyancy forces are large [4, 5, 10, 23]. Numerous experiments have explored the interaction between stratification and wake evolution with close observation to the generation of internal gravity waves and coherent structures [19, 37]. Direct Numerical Simulation (DNS) provides further insight into the physics of the flow, particularly with its turbulence properties [25, 27]. Background turbulence increases the turbulent kinetic energy and energy transfer in the wake, which in turn lowers the mean velocity and increases horizontal spreading [26]. Excess momentum results in increased turbulent kinetic energy and qualitative changes in the wake dynamics, particularly in downstream vortical structures [6]. High levels of stratification in the wake create a non–equilibrium region in which the mean velocity decay is reduced [28, 36].

Originally studied as a disc–with–center–jet [24] and later with self–propelled axisymmetric bodies [18, 33], the net–zero–momentum wake functions as a theoretical model of a self–propelled marine vehicle. Beyond experiment, the study of self–propelled wakes includes several numerical methods. Ordered by increasing fidelity and computational expense these methods include [32]: panel/lattice methods, actuator models, and fully resolved rotating geometry. Generalized actuator models include the Actuator Disk (AD), Actuator Line (AL), and Actuator Surface (AS) models. Each of these models impose a body force over a volume in a Computational Fluid Dynamics (CFD) simulation to simulate the effects of a propeller on the surrounding fluid. Although a fully resolved propeller may offer more fidelity, its computational requirements are often large, so an actuator model provides a cost–effective alternative [16].

In a self–propelled near wake, the mixed–patch structure and overall potential energy depend largely on the propulsor. A single propeller will mix fluid unopposed within the swirling region of the wake. Contra–rotating propellers of equivalent thrust, will modify the initial swirl profile thereby changing the shape of the downstream mixed patch and reducing its
potential energy. Contra–rotating propeller blades add additional complexity to the interaction between root and tip vortices and reduce the swirling kinetic energy of the wake. These influences on the near wake may be compared to the simplified case of a zero–swirl, jet–propelled configuration with uniform–velocity, which results in the smallest generation of the potential energy in the wake.

The present study is an extension of Jones and Paterson [17]. The unsteady Reynolds–Averaged Navier–Stokes (RANS) equations are solved to examine the near–wake evolution of the stratified, turbulent, net–zero–momentum propeller wake of the axisymmetric Iowa Body using three different propulsion schemes: single propeller, dual contra–rotating propellers (CRP), and a zero–swirl, uniform–velocity jet. The propellers are simulated using the AL model. The Iowa Body hull geometry is chosen for comparison to the non–stratified experiment of Hyun and Patel [13], which is the only known experiment to have phase–averaged propeller data for a self–propelled axisymmetric body. The authors have previously shown good agreement to this experiment for towed and self–propelled configurations [16]. Flow visualization reveals the interaction between propeller–root and tip vortices and the additional complexity introduced by CRP. Comparison between instantaneous and time–averaged cross–plane profiles demonstrates the transition from near– to far–wake regions. Circumferentially–averaged profiles of velocity reveal the evolution of momentum, with observations drawn in comparison to the theoretical disc–with–center–jet that is often used in far–wake simulations [10]. Mixed–patch velocity and temperature–deviation cross–plane profiles show the structure of kinetic and potential energy in the developed wake. Finally, the relative growth, decay, or persistence of integrated kinetic and potential energy of each propulsion scheme is considered. Compared to the single–propeller configuration, the CRP configuration is more effective at reducing potential and swirling kinetic energy in the wake, with potential energy reductions similar to that of the zero–swirl jet.

4.2 Approach

4.2.1 Governing Equations

This fluid–flow problem is defined by the unsteady Reynolds–averaged Navier–Stokes (RANS) equations in Boussinesq form with an additional body force term \( f_p \) to account for the propeller model.

\[
\frac{\partial U_i}{\partial t} = 0
\]

\[
\frac{\partial U_i}{\partial t} + \frac{\partial (U_i U_j)}{\partial x_j} = -\frac{1}{\rho_0} \frac{\partial \rho}{\partial x_i} + \nu \frac{\partial^2 U_i}{\partial x_j \partial x_j} + \frac{\partial}{\partial x_j} u_i^r u_j^r + \frac{\Delta \rho}{\rho_0} g_j \delta_{ij} + \frac{1}{\rho_0} f_p
\]  

(4.2)

The equations are written in terms of the non–inertial velocity \( U_i \). In the equations, \( t \) is time, \( \nu \) is the kinematic viscosity, and \( \rho \) is density. The density is expressed as \( \rho = \rho_0 + \Delta \rho \), where
\( \rho_0 \) is a reference value and \( \Delta \rho \) is the deviation from that value. The gravitational vector \( g_j \) points downward in the negative \( z \) direction, where \( z \) is the upward–positive, vertical position. This formulation includes the piezometric pressure, \( \hat{p} = p - \rho_0 g z \) where \( g \) is the magnitude of the gravitational vector.

The governing equations are solved using a custom solver written with the CFD framework OpenFOAM. This custom solver takes into account salinity and temperature transport and the corresponding turbulent fluctuations. The transport of temperature \( T \) and salinity \( S \) in the stratified environment are determined through the following equations with the diffusion coefficients \( \kappa_T \) and \( \kappa_S \).

\[
\frac{\partial T}{\partial t} + \frac{\partial (U_j T)}{\partial x_j} = \kappa_T \frac{\partial^2 T}{\partial x_j \partial x_j} + \frac{\partial}{\partial x_j} \overline{u'_j u'} \quad (4.3)
\]

\[
\frac{\partial S}{\partial t} + \frac{\partial (U_j S)}{\partial x_j} = \kappa_S \frac{\partial^2 S}{\partial x_j \partial x_j} + \frac{\partial}{\partial x_j} \overline{u'_j s'} \quad (4.4)
\]

The Reynolds stresses \( \overline{u'_i u'_j} \) and turbulent fluxes \( \overline{u'_j T'} \) and \( \overline{u'_j S'} \) are determined using a linear eddy–viscosity closure model.

\[
-\overline{u'_i u'_j} = 2\nu_t S_{ij} - \frac{2}{3} k \delta_{ij} \quad (4.5)
\]

\[
-\overline{u'_j T'} = \frac{\nu_t}{\sigma_T} \frac{\partial T}{\partial x_j} \quad (4.6)
\]

\[
-\overline{u'_j S'} = \frac{\nu_t}{\sigma_S} \frac{\partial S}{\partial x_j} \quad (4.7)
\]

In these equations \( \nu_t \) is the eddy viscosity, \( S_{ij} \) is the mean rate of strain, and \( k \) is the turbulent kinetic energy. For this study, the \( k - \omega \) SST turbulence model is chosen to compute \( \nu_t \) due to its ease of implementation and relative advantage in computing the attached flow over a body [22]. Production terms in the \( k - \omega \) equations are modified to include buoyancy effects, but in the near wake they are small in comparison to the production due to shear. Wall functions are used in the computation of \( k \) and specific turbulence dissipation \( \omega \) at wall boundaries to relax mesh requirements near the hull in the high Reynolds–number flow.

Density is computed by solving the UNESCO seawater equation of state [9]. For the given problem, it is appropriate to approximate the secant bulk modulus as constant at sea–level conditions, even though it is a function of salinity, temperature, and pressure. Thus, the secant bulk modulus is \( K(S, T, p) = K(0, 20, p_{atm}) \) where \( p_{atm} \) is atmospheric pressure. Additionally, substituting the hydrostatic pressure for the total pressure, the equation of
Chapter 4. Influence of Propulsion Type on the Stratified Near Wake of a Self-Propelled Body

The state becomes,

\[ \rho(S, T, p) = \frac{\rho(S, T, 0)}{1 - p/K(S, T, p)} \]  
\[ \rho(S, T, 0) = \left( a_0 + a_1 T + a_2 T^2 + a_3 T^3 + a_4 T^4 + a_5 T^5 \right) \]
\[ + (b_0 + b_1 T + b_2 T^2 + b_3 T^3 + b_4 T^4) S \]
\[ + (c_0 + c_1 T + c_2 T^2) S^{3/2} \]
\[ + d_0 S^2 \]  
(4.8)

The \( a_n, b_n, c_n \) and \( d_0 \) terms are empirical coefficients given in Table 4.1. Because the environment in the present study is isohaline, only changes in temperature from the thermally-stratified background affect changes in density.

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>Value</th>
<th>Coefficient</th>
<th>Value</th>
<th>Coefficient</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( a_0 )</td>
<td>( 9.998425 \times 10^{+2} )</td>
<td>( b_0 )</td>
<td>( 8.2449 \times 10^{-1} )</td>
<td>( c_0 )</td>
<td>( -5.7247 \times 10^{-3} )</td>
</tr>
<tr>
<td>( a_1 )</td>
<td>( 6.793952 \times 10^{-2} )</td>
<td>( b_1 )</td>
<td>( -4.0899 \times 10^{-3} )</td>
<td>( c_1 )</td>
<td>( 1.0227 \times 10^{-4} )</td>
</tr>
<tr>
<td>( a_2 )</td>
<td>( -9.095290 \times 10^{-3} )</td>
<td>( b_2 )</td>
<td>( 7.6438 \times 10^{-5} )</td>
<td>( c_2 )</td>
<td>( -1.6546 \times 10^{-6} )</td>
</tr>
<tr>
<td>( a_3 )</td>
<td>( 1.001685 \times 10^{-4} )</td>
<td>( b_3 )</td>
<td>( -8.2467 \times 10^{-7} )</td>
<td>( d_0 )</td>
<td>( 4.8314 \times 10^{-4} )</td>
</tr>
<tr>
<td>( a_4 )</td>
<td>( -1.120083 \times 10^{-6} )</td>
<td>( b_4 )</td>
<td>( 5.3875 \times 10^{-9} )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( a_5 )</td>
<td>( 6.536332 \times 10^{-9} )</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

4.2.2 Kinetic and Potential Energy

The evolution and transfer of energy in the wake is examined in the form of kinetic and potential energy defined as,

\[ ke = \frac{1}{2} \rho U^2, \quad pe = -\frac{1}{2} \frac{g}{\partial \rho_0/\partial z} (\rho - \rho_0)^2 \]  
(4.10)

\[ KE = \iiint_A ke \, dA, \quad PE = \iiint_A pe \, dA, \]  
(4.11)

The per-unit-volume energy \( ke \), and \( pe \) may be integrated over an axial slice of area \( A \) in the wake to find energy per-unit-length, \( KE \) and \( PE \), as functions of downstream distance. Kinetic energy is computed for the magnitude of velocity and also individually for each component of velocity in cylindrical coordinates. The potential energy per-unit-volume \( pe \) follows the formulation of Holliday and McIntyre [11].
4.2. Approach

4.2.3 Actuator–Line Model

The unsteady propeller for each non–BOR hull form is simulated using an AL model from the Simulator for Wind Farm Applications (SOWFA) library [2]. The AL model projects a distributed line of force \( f_p \) in the place of each propeller blade,

\[
f_p(r) = \frac{F_p}{\varepsilon^{3.5/2}} \exp \left[ -\left( \frac{r}{\varepsilon} \right)^2 \right]
\]

(4.12)

where \( F_p \) is the actuator element force composed of contributions from lift \( F_L \) and drag \( F_D \). The distance between CFD cell center and actuator point is \( r \), and \( \varepsilon \) controls the Gaussian width. This function decays to 1% of its maximum value when \( \varepsilon = 2.15r \). If \( \varepsilon \) is too small, numerical oscillations arise, and if \( \varepsilon \) is too large, the applied body forces will be smoothed considerably. Troldborg [39] recommends \( \varepsilon = 2\Delta x \) where \( \Delta x \) is the grid spacing at the actuator position. Martinez et al. [21] developed best practices for AL modeling and suggested \( \varepsilon > 2\Delta x \). For the present study, \( \varepsilon = 4\Delta x \) was selected because it eliminated the numerical instabilities that arose when \( \varepsilon = 2\Delta x \) was assigned. The Cartesian–mesh region of the propeller was refined to a resolution such that, \( \Delta x/\Delta b = 0.74 \), where \( \Delta b \) is the width of each hydrofoil section. Martinez et al. [21] suggests a value smaller than 0.75. Lift and drag at each section are computed from a lookup table of lift and drag coefficients \( C_L \) and \( C_D \) as functions of \( \alpha \),

\[
F_L = \frac{1}{2}C_L(\alpha)\rho U_{rel}^2 c w, \quad F_D = \frac{1}{2}C_D(\alpha)\rho U_{rel}^2 c w
\]

(4.13)

where \( \rho \) is the density, \( U_{rel} \) is the local flow speed, \( c \) is the chord and \( w \) is the width of the actuator section. The relationship between \( C_L \) and \( C_D \) with \( \alpha \) must be predetermined from experiment, simulation, or theory for each hydrofoil section. Figure 4.1 shows the magnitude of the projected propeller body force \( |f_p|/(\rho_0 R_p \text{ rps}^2) \) on the AL propeller plane of the single–propeller case, where \( R_p \) is the propeller radius and rps is the propeller rotations per second.

4.2.4 Iowa Body

The axisymmetric Iowa Body, described in the experiment of Hyun and Patel [13], is shown in Figure 4.2 for the standard, single–propeller case. This geometry is representative of a typical marine vehicle without appendages. Features of this geometry are listed in Table 4.2 where \( L \) is the body length, \( D \) is the body diameter, \( D_p \) is the propeller diameter, and \( D_h \) is the hub diameter.

Minor modifications are made to the Iowa Body hull for the CRP and jet configurations. For the CRP configuration, the hub is extended by the length of the rotating portion so that a second propeller may be placed directly downstream of the first. For the jet configuration, the hub is truncated at the propeller plane to function as an exhaust port. In effect, the zero–swirl, uniform–velocity jet exhausts with an initial diameter of \( D_h \).
Chapter 4. Influence of Propulsion Type on the Stratified Near Wake of a Self–Propelled Body

Figure 4.1: Non–dimensional body force on mesh slice at propeller plane $||f_p||/(\rho_0R_p \text{ rps}^2)$ for the single–propeller case.

Figure 4.2: Standard Iowa Body profile.

Table 4.2: Iowa Body geometry.

<table>
<thead>
<tr>
<th>Feature</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$L/D$</td>
<td>10.90</td>
</tr>
<tr>
<td>$D/D_p$</td>
<td>1.369</td>
</tr>
<tr>
<td>$D_p/D_h$</td>
<td>6.266</td>
</tr>
<tr>
<td>Hub location</td>
<td>$0.9688 &lt; x/L &lt; 0.9832$</td>
</tr>
<tr>
<td>Propeller location</td>
<td>$x/L = 0.9755$</td>
</tr>
<tr>
<td>Number of blades</td>
<td>3</td>
</tr>
<tr>
<td>Propeller hydrofoil</td>
<td>NACA 66–Modified</td>
</tr>
</tbody>
</table>
4.2.5 Iowa Body Propeller

The Iowa Body propeller is defined by 36 discrete sections to account for variations in radial propeller–blade geometry. Sectional lift $C_\ell$ is computed using the analytic expression of Brockett [3] for the NACA 66–Modified foil,

$$C_\ell = 2\pi (1 - 0.83\tau)(\alpha + 2.05f)$$

where $\alpha$ is the local flow angle of attack, $\tau$ is the maximum thickness ratio, and $f$ is the maximum camber ratio. Sectional drag $C_d$ is imposed by combining viscous and induced drag at each section,

$$C_d = C_{d0} + \frac{C_\ell^2}{\pi e AR}$$

where $C_{d0}$ is the viscous drag, $e$ is the efficiency factor, and $AR$ is the aspect ratio.

For the present unsteady simulations, $\alpha$ at each section of the propeller blades remains below 3° at every instant in time. Because $\alpha$ remains small, these analytic expressions do not require additional conditions for stall. Pitch, chord, thickness, and camber distributions for the Iowa Body propeller blade are tabulated in Hyun [12]. The Iowa Body propeller has zero rake and zero skew.

4.2.6 Computational Mesh

The three computational meshes were generated using the software $cfMesh$ [1]. Cells are focused near the body, the propulsor region, and in the wake. The hull is located at a depth of one body length. The inlet, outlet, and far–field boundaries are located two body lengths away from the hull. Comparison to simulations from spatially–larger meshes showed that the boundaries of the computational domain did not affect the solution. Mesh design and quality features are listed in Table 4.3. Because wall functions are used in the computation of turbulence variables, the dimensionless wall distance requirement of $y^+ < 100$ can capture the boundary layer effects and the viscous drag of the hull even for boundary cells where $y^+ \approx 100$. A grid–refinement study of the propeller– and wake–region cells showed that 100 cells/$D_p$ adequately resolved the AL model and downstream wake cross–plane profiles. These meshes are also visualized in Figure 4.3. Cutting planes reveal the distribution of cells surrounding the hull and in the wake region. Views of the propulsor region show how the mesh is modified for each configuration. A single AL–modeled propeller is implemented within the highlighted region for the single–propeller case. For the CRP configuration, the hub is extended with one AL–modeled propeller placed behind the first. There is no AL model for the jet configuration since there is no propeller. Instead, fluid is exhausted from the truncated hub.
Table 4.3: Mesh design and quality features.

<table>
<thead>
<tr>
<th>Mesh feature</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Boundary layer cells</td>
<td>&gt; 20</td>
</tr>
<tr>
<td>Near–wall mesh spacing</td>
<td>$y^+ &lt; 100$</td>
</tr>
<tr>
<td>Propulsor and wake cells/$D_p$</td>
<td>100</td>
</tr>
<tr>
<td>Wake region extends to</td>
<td>$x/L = 1.6$</td>
</tr>
<tr>
<td>Total number of cells</td>
<td>$2 \times 10^7$</td>
</tr>
<tr>
<td>Maximum aspect ratio</td>
<td>AR &lt; 170</td>
</tr>
<tr>
<td>Maximum non–orthogonality</td>
<td>&lt; 45°</td>
</tr>
<tr>
<td>Maximum skewness</td>
<td>&lt; 0.8</td>
</tr>
</tbody>
</table>

(a) Vertical cutting plane through standard Iowa Body mesh with nested refinement.
(b) Boundary–layer cells.
(c) Propulsor region for standard mesh.
(d) Propulsor region for CRP mesh.
(e) Propulsor region for jet mesh.

Figure 4.3: Computational meshes generated for each configuration.
4.2. Approach

4.2.7 Numerical Methods

The Navier–Stokes unsteady mass and momentum equations are solved by using the Pressure–Implicit with Splitting of Operators (PISO) method [14]. This segregated approach decouples operations on pressure and velocity variables. At each time step, the following procedure is followed in the customized OpenFOAM solver. First, the momentum equations are solved to provide velocity by using pressure from the previous time step. Next, the pressure–Poisson equation is solved iteratively with corrections to velocity to conserve mass. Three inner iterations are used in the present study, each with an additional mesh non–orthogonality correction step. After completion of these inner iterations, turbulence quantities are solved for, followed by salinity and temperature. The time step is then advanced.

Implicit, second–order, backward differencing is used in temporal discretization, while the cell–centered finite volume method is used in spatial discretization. A second–order, linear–upwind scheme is applied to the advective term of the momentum equations. A first–order, upwind scheme is applied to turbulence quantities, and a second–order, linear scheme is applied to all other divergence terms. Laplacian terms are discretized using a second–order, linear scheme that is partially–limited to correct for mesh non–orthogonality.

Two iterative methods are employed to solve the resulting systems of algebraic equations. The pressure equation is solved using the Preconditioned Conjugate Gradient (PCG) method with a residual tolerance of $10^{-6}$. The momentum, scalar transport, and turbulence equations are solved using the Pre–Conditioned Bi–Conjugate Gradient (PBiCG) scheme with a residual tolerance of $10^{-8}$.

4.2.8 Initial and Boundary Conditions

Several boundary conditions are employed. Velocity at the inlet is set to the freestream velocity $U_0$ through a Dirichlet boundary condition. The no–slip condition is set on the hull boundary, and the slip condition is set in the far field. Zero–gradient conditions are specified for velocity and pressure in the outlet. Background turbulence values of $k$ and $\omega$ are computed assuming a turbulence intensity of 1% and eddy viscosity ratio $\nu_t/\nu$ of 100. Turbulence variables on the hull boundary are computed with wall functions. Other variables satisfy the zero–gradient Neumann boundary condition.

Initial conditions for pressure and velocity are computed by solving the potential flow equations. The PISO algorithm is then used in the transient simulation. Distributed body forces from the propeller–blades rotate at each time step at the propeller’s rotation rate. The simulation is then run until initial–transient flow features advect far downstream and a periodic wake flow field is found.
4.2.9 Flow Field Analysis

This study examines primary flow variables including: deviation of temperature from the background $\Delta T$ and the axial, radial, and azimuthal velocities $U_x$, $U_r$, and $U_\theta$. The second invariant of the velocity gradient tensor $Q$ is computed to visualize vortical structures. The cross–plane–integrated kinetic and potential energies are examined, where the kinetic energy is considered exclusively for each of the three components of velocity $KE_x$, $KE_r$, and $KE_\theta$. Data is extracted in axial planes between $0.9755 \leq x/L \leq 1.5$ where $x$ is the downstream distance from the bow of the body and $L$ is the body length.

4.2.10 Flow Coefficients and Case Studies

Several of the important flow coefficients for this propeller–driven flow are the Reynolds number $Re_L$, advance ratio $J$, thrust coefficient $C_T$, and torque coefficient $C_Q$. An alternate expression for the thrust coefficient $C_T^*$ is computed for comparison to the jet configuration.

$$Re_L = \frac{U_0 L}{\nu}, \quad J = \frac{U_0}{nD_p}, \quad C_T = \frac{F_T}{\rho_0 n^2 D_p^3}, \quad C_T^* = \frac{F_T}{\frac{1}{2} \rho_0 U_0^2 \pi R^2}, \quad C_Q = \frac{T_Q}{\rho_0 n^2 D_p^5} \quad (4.16)$$

For these expressions, $U_0$ is the freestream velocity, $\nu$ is kinematic viscosity, $D_p$ is the diameter of the propeller, $R$ is the radius of the Iowa Body, $n$ is the propeller speed in revolutions per second, $F_T$ is the thrust, and $T_Q$ is the torque. The thrust–to–drag ratio is $F_T/F_D$. The Reynolds number for this study is $Re_L = 310 \times 10^6$, a typical operating condition in the ocean. Other coefficients are listed in Table 4.4. The fore and aft propellers are listed individually for the CRP case, and total thrust is equivalent for all cases.

<table>
<thead>
<tr>
<th>Configuration</th>
<th>$J$</th>
<th>$C_T$</th>
<th>$C_T^*$</th>
<th>$C_Q$</th>
<th>$F_T/F_D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Single</td>
<td>0.86</td>
<td>0.047</td>
<td>0.084</td>
<td>0.011</td>
<td>0.99</td>
</tr>
<tr>
<td>CRP (fore)</td>
<td>0.90</td>
<td>0.024</td>
<td>0.041</td>
<td>0.0071</td>
<td>0.50</td>
</tr>
<tr>
<td>CRP (aft)</td>
<td>0.86</td>
<td>0.023</td>
<td>0.041</td>
<td>0.0072</td>
<td>0.50</td>
</tr>
<tr>
<td>Jet</td>
<td>-</td>
<td>-</td>
<td>0.082</td>
<td>0</td>
<td>1.07</td>
</tr>
</tbody>
</table>

The Froude number $Fr$ provides a measure of the density stratification, where an infinite $Fr$ means zero stratification, and a small $Fr$ means high levels of stratification. The present study considers a linearly varying temperature stratification, typical of an ocean environment, with a Froude number of $Fr = 350$, where,

$$Fr = \frac{1}{N} \frac{U_0}{N D}, \quad \text{and} \quad N = \frac{1}{2\pi} \sqrt{-g \frac{\partial \rho}{\partial z}}. \quad (4.17)$$
4.3. Results

In these expressions $N$ is the Brunt Väisälä frequency, $g$ is acceleration due to gravity, and $z$ is the vertical coordinate. The influence of buoyancy on the near–wake fluid dynamics is often small and can be quantified by the Richardson number $Ri$, which is the ratio of buoyancy to flow gradient terms [19].

$$Ri = \frac{g}{T_0} \frac{dT}{dz} \left( \frac{dU}{dz} \right)^2$$

(4.18)

Here, $T_0$ is a reference temperature, $T$ is the mean temperature, and $U$ is the mean velocity. For the single–propeller case, $Ri \approx 2.54 \times 10^{-3}$ which indicates that the near–wake inertial forces of the propeller dominate the buoyancy forces. As the local velocity $U_x$ decays, $Ri$ increases and buoyancy forces become important further downstream in the far wake, beyond the geometric bounds of these simulations.

4.3 Results

4.3.1 Near–Wake Transition

Individual vortices are visualized using the second invariant of the velocity gradient tensor $Q$. Figure 4.4 shows contour surfaces of the non–dimensionalized $(L/U_0)^2Q = 16.9$ with a vertical cutting plane colored by axial velocity defect $U_x/U_0 - 1$ that extends to half of a body length downstream where $x/L = 1.5$. For the single–propeller case, root and tip vortices induced by the propeller are apparent. These vortices follow a helical path and disappear by $x/L \approx 1.25$. For the CRP case, additional complexity is introduced by the interaction between the two propellers. Complicated vortical structures are visible until $x/L \approx 1.35$. Negligible vortical structures are found in the case of the zero–swirl jet. This figure illustrates the complexity introduced by contra–rotating propellers in comparison to the other two cases.

To better understand the transition from near to far wake regimes in the propeller–driven cases, axial planes behind the propulsor are examined. Figure 4.5 compares the instantaneous and time–averaged axial velocity defect field $U_x/U_0 - 1$ for the single propeller case. Instantaneous fields are taken at an arbitrary time long after initial–transient features have disappeared from the simulation and the flow field has become periodic. Time–averaging occurs over temporal interval of two periods of the propeller. Near the propellers, at $x/L = 1.01$ and $x/L = 1.10$, individual propeller–blade wakes can be seen in the instantaneous field. These blade wakes follow the azimuthal motion of the propellers directly upstream. The time–averaged field, by contrast, is axisymmetric. Further downstream at $x/L = 1.3$ only a small variation is seen between the instantaneous and time–averaged fields, and by $x/L = 1.45$ the two contour maps are nearly identical showing that the wake is steady and axisymmetric. By half of a body length downstream, the flow is stationary in time and space when viewed from a body–fixed reference frame.
Figure 4.4: Flow visualization for $x/L \leq 1.5$ using $Q$-criterion visualization non-dimensionalized as $(L/U_0)^2Q = 16.9$ colored by $U_x/U_0 - 1$ with vertical cutting plane through mesh.
4.3. Results

For the CRP case shown in Figure 4.6, similar observations are drawn. Unsteadiness is apparent for \( x/L \leq 1.10 \), however by \( x/L = 1.45 \) the cross–plane profile is temporally stationary. In this case, a unique hexagonal shape is formed due to the interaction between the two opposing three–bladed propellers. This shape is still present at half of a body length downstream of the propulsor.

![Figure 4.5: Instantaneous (top) and time–averaged (bottom) velocity defect \( U_x/U_0 - 1 \) for the single propeller case.](image)

### 4.3.2 Velocity Profiles

The evolution of the near wake may also be described by circumferentially–averaged velocity profiles. Figure 4.7 shows the circumferentially–averaged velocity defect profiles \( U_x/U_0 \) for the three self–propelled configurations at the downstream positions \( x/L = 1.01, x/L = 1.3, \) and \( x/L = 1.5 \). Just behind the propulsor at \( x/L = 1.01 \), the jet is shown to have uniform, positive velocity leaving the exhaust port, while negative velocity due to drag appears for \( r/R_p > 0.2 \). By \( x/L = 1.3 \) and further at \( x/L = 1.5 \), the jet profile appears as a classical net–zero–momentum wake and may be described using the analytical formulation of the disc–with–center–jet.

The propeller–driven cases, show their own unique profiles. Positive momentum from the propellers exists in a region near \( r/R_p \approx 0.6 \), while negative momentum due to drag from the body exists near the center and further outward. These two circumferentially–averaged profiles are nearly equivalent and decay at similar rates, which is explained by the similar distributions of axial momentum. They also may be defined analytically using a process
Chapter 4. Influence of Propulsion Type on the Stratified Near Wake of a Self–Propelled Body

Figure 4.6: Instantaneous (top) and time–averaged (bottom) velocity defect $U_x/U_0 - 1$ for the CRP case.

described in Jones and Paterson [16]. For all configurations, the positive momentum decays more quickly than the negative momentum, which is a feature of idealized wakes described by Tennekes and Lumley [38]. The theoretical, axisymmetric drag wake decays with the power of -2/3, while the theoretical, axisymmetric jet decays with the power of -1.

Circumferentially–averaged profiles of the swirl component of velocity $U_\theta$ are shown in Figure 4.8. The jet configuration profile exhibits zero swirl because there are no sources of swirl for this case. The jet exhaust contains uniform axial velocity, the body is axisymmetric, and buoyancy forces are relatively small. The single–propeller configuration at $x/L = 1.01$, shows swirl imparted by the propeller and rotating hub. By $x/L = 1.3$ and $x/L = 1.5$, most of the momentum due to swirl exists in a region centered at $r/R_p \approx 0.4$. In the case of the CRP, regions of positive and negative swirl develop due to interference between the opposing, contra–rotating blades. Throughout the near wake, the CRP swirl magnitude is attenuated, remaining less than half of that of the single propeller, which is explained by the interactions of opposing azimuthal forces of the two propellers.

4.3.3 Velocity and Temperature Fields in the Mixed–Patch

The wake is mixed by half of a body length downstream of the stern. Unsteadiness from the propulsor has disappeared and axial gradients are small in comparison to the transverse. This location is significant because a cross–plane profile may be considered as the initial conditions for further far–field simulations, which is beyond the scope of this paper. Unique
4.3. Results

Figure 4.7: Circumferentially–averaged velocity defect profiles for each configuration at various distances downstream.

Figure 4.8: Circumferentially–averaged swirl velocity profiles for each configuration at various distances downstream.
features of the cross–plane profiles of velocity and temperature deviation are presented.

**Velocity Field**

Cross–plane contour maps of the axial velocity defect $U_x/U_0 - 1$ at $x/L = 1.5$ are shown for the single propeller and CRP cases in Figure 4.9. For the single propeller, axial velocity is axisymmetric and has previously been fit to an analytical curve as a function of radial distance [16]. The CRP, however, is not axisymmetric, and a steady, hexagonal profile is formed. The geometric shape is attributed to the two three–bladed propellers interacting with one another. Unlike the single–propeller profile, the CRP profile is a function of both radial and azimuthal positions. To create an analytical expression as a function of radial position alone, the profile must first be circumferentially–averaged.

![Figure 4.9: Stationary velocity defect profiles at $x/L = 1.5$.](image)

The swirl component of velocity $U_\theta/U_0$ is shown in Figure 4.10. Again, the single–propeller velocity is axisymmetric, while the CRP velocity has discernible geometry. The magnitude of swirl velocity is much higher for the single propeller case, since the CRP propulsor imparts opposing azimuthal forces. Swirl velocity from the CRP is less than half of that of the single propeller and varies in azimuthal sign.

**Temperature Field**

Because these simulations take place in a thermally–stratified environment, any vertical redistribution of the fluid will generate potential energy. Mixing in the wake plays an im-
4.3. Results

Figure 4.10: Stationary swirl velocity profiles at $x/L = 1.5$.

important role in the redistribution of temperature $T$. Given that the background temperature field $T_b$ is initially linearly stratified, mixing from the wake develops a temperature deviation $\Delta T = T - T_b$. This field is non–dimensionalized by the linear change in temperature over the depth of one propeller–blade length $\Delta T_{R_p}$.

Figure 4.11 shows $\Delta T/\Delta T_{R_p}$ for the single–propeller case. Additional radial profiles help to visualize how the field varies in polar coordinates. A unique cross–plane profile shape is formed that is steady in time. Colder fluid has been driven to the top, while warmer fluid has been driven to the bottom of the wake. A maximum is found near the center of the warm region and a minimum is found near the center of the colder region. Two “tails” are shown trailing off of the warm and cold regions as a result of the counter–clockwise swirling motion of the fluid due to the propeller.

The mixed–patch $\Delta T/\Delta T_{R_p}$ cross–plane profile of the CRP case is shown in Figure 4.12. Compared to the single–propeller case, the magnitude of $\Delta T/\Delta T_{R_p}$ is less than half. The profile is split between an inner region where clockwise–swirling fluid dominates and an exterior region where counter–clockwise–swirling fluid dominates as shown previously in Figure 4.8. The complexity and lower $\Delta T/\Delta T_{R_p}$ magnitude in the profile arise directly from the initial interactions of the opposing azimuthal forces of the contra–rotating blades that drive the swirling fluid. While a single propeller can transport the temperature field across the swirling wake region unimpeded, the addition of an opposing propeller directly counters this effect. The net–swirl in the wake is reduced and regions of both positive and negative swirl exist. The inner, negative–swirl region forms a $\Delta T/\Delta T_{R_p}$ profile that mirrors the single–propeller case because of the sign difference in swirl. The outer, positive swirl
Chapter 4. Influence of Propulsion Type on the Stratified Near Wake of a Self–Propelled Body

106

Figure 4.11: Single propeller $\Delta T/\Delta T_{R_p}$ at $x/L = 1.5$.

region shares the same sign of $\Delta T/\Delta T_{R_p}$ as the single–propeller case. In effect, the net loss in swirl reduces the overall potential energy.

Finally, the mixed–patch $\Delta T/\Delta T_{R_p}$ cross–plane profile of the jet case is shown in Figure 4.13. For this configuration, the magnitude of $\Delta T/\Delta T_{R_p}$ is the smallest due to the absence of swirl. Instead, the shearing of axial momentum and potential effects from the body control the shape of the profile. The positive–momentum “jet” core entrains fluid from the negative–momentum “drag” periphery of the wake. Potential effects from the body further influence the temperature distribution. The resulting transport of $\Delta T/\Delta T_{R_p}$ suspends warmer fluid above colder fluid in the center, and the reverse in sign in the periphery. Without swirl, the distribution of $\Delta T/\Delta T_{R_p}$ in the central core of the jet wake is opposite in sign to that of the single–propeller case.

Comparison to a Perfectly–Mixed Temperature Field

The simulated cross–plane profiles may be compared to the idealized, perfectly–mixed profile shown in Figure 4.14. This conceptual profile assumes perfect mixing such that the temperature distribution $T$ is uniform up until the boundary of the wake disc, beyond which, $T = T_b$. The temperature deviation $\Delta T = T - T_b$, however, varies because of changes from the background stratification within the disc. Given the linear background stratification, $\Delta T$ also varies linearly in the vertical direction. The single propeller case relates most closely to this idealized profile, but because it does not perfectly–mix $T$, differences can be observed. The upper and lower regions of colder and warmer fluid are shifted from the centerline, and the
4.3. Results

(a) Cross–plane contours.  
(b) Profile in radial directions.

Figure 4.12: CRP $\Delta T/\Delta T_{R_p}$ at $x/L = 1.5$.

(a) Cross–plane contours.  
(b) Profile in radial directions.

Figure 4.13: Jet configuration $\Delta T/\Delta T_{R_p}$ at $x/L = 1.5$. 
maximum temperature deviations are not on the wake boundary but instead closer inward. Results from the single propeller case show that $T$ is not mixed uniformly within the disc of the swirling wake.

![Cross-plane contours](image1)

(a) Cross–plane contours.

![Profile in radial directions](image2)

(b) Profile in radial directions.

Figure 4.14: Idealized mixed–patch profile of $\Delta T/\Delta T_{R_p}$ for a wake region of constant $T$.

### 4.3.4 Potential and Kinetic Energy Evolution

Kinetic and potential energy is integrated along axial planes downstream for the three configurations as shown in Figure 4.15. Kinetic energy $KE$ is computed individually for the three components of velocity: radial, swirl, and axial as $KE_r$, $KE_\theta$, $KE_x$, as well as for the velocity magnitude as $KE$. Downstream distance is described both by $x/L$ measured from the bow and by $x'/D_p$ measured from the stern.

For all three cases, $KE_r$ decays more rapidly than $KE_x$ and $KE_\theta$. The swirl component notably exhibits the slowest decay in the near wake, an observation consistent with Sirviente and Patel [35]. For the single–propeller case, this relative persistence leads to a rise in $PE$ due to expansion of the wake and entrainment of the surrounding passive scalar $T$, indicating a change in density. For the CRP, the $PE$ does not grow due to the opposing regions of positive and negative swirl velocity. Instead the $PE$ decays with a rate similar to the zero–swirl jet. This result shows that the contra–rotating blades can effectively reduce $PE$.

Comparing the swirl component for the three cases, $KE_\theta$ of the CRP is an order–of–magnitude lower than that of the single propeller. Counteracting azimuthal momentum
4.3. Results

Figure 4.15: Integrated energy evolution downstream of the vehicle for each configuration, with $x$ measured from the bow of the hull and $x'$ measured from the stern.
leads to a reduction in the swirl kinetic energy. The reduced $KE_\theta$ of the CRP is consistent with its reduced $PE$. The jet $KE_\theta$ is small because of the initial absence of swirl.

Additionally, comparison of the total $KE$ shows that the CRP is the most–effective configuration for the total reduction of energy by $x/L = 1.5$. This result suggests that the CRP velocity field decays most–rapidly. The single propeller is less effective than the jet due to persisting $KE_\theta$ from its unidirectional swirling velocity.

### 4.4 Conclusions

The influence of swirl on the evolution of self–propelled, stratified near wakes and the development of the mixed patch has not previously been well–characterized. In this study, the linearly stratified near wake of the Iowa body was investigated with three separate propulsor configurations: single propeller, contra–rotating propellers, and a zero–swirl, uniform–velocity jet. Unsteady, rotating propeller blades were simulated using an AL model in an unsteady RANS computation. Comparison between the configurations revealed unique differences in the evolution of the near wake.

While clear root and tip vortices were visible in the single–propeller case, the CRP disrupted these structures, introducing additional complexity in the wake evolution. Nevertheless, by half of a body length downstream, the wake flow fields were steady in time. The single–propeller and CRP cases shared similar circumferentially–averaged axial velocity defect profiles due to similar spanwise loading in the propulsor. Swirl velocity, however varied between the two propeller–driven cases, with the CRP introducing both positive and negative swirl regions exhibiting half of the magnitude of the single–propeller case. Furthermore, by half of a body length downstream, the magnitude of the temperature deviation $\Delta T/\Delta T_{R_p}$ for the CRP was less than half of that of the single propeller. The jet $\Delta T/\Delta T_{R_p}$ magnitude was the smallest, due to the absence of swirl. Contour maps of velocity revealed that the single propeller has an axisymmetric profile, whereas the CRP exhibits a unique hexagonal structure as a result of its two three–bladed propellers. The evolution of kinetic and potential energy varied as a direct result of the swirl imparted by each propulsor. Because of the interaction of positive and negative swirl, the CRP configuration showed an order–of–magnitude lower swirling kinetic energy compared to the single propeller configuration. Additionally, its potential energy was similar in decay and magnitude to that of the swirl–free jet, and the total kinetic energy decayed most rapidly out of the three propulsion schemes. These results indicate that the CRP can effectively reduce potential energy that would otherwise develop from a single–propeller configuration.
Bibliography


Chapter 5

Influence of Hull–Form on the Stratified Near Wake of a Self–Propelled Body

(Work adapted from Jones and Paterson [17].)

Abstract

This study investigates the influence of hull–form variation on a self–propelled body in a linearly–stratified environment typical of the ocean. The Reynolds number is $Re_L = 310 \times 10^6$, based on the freestream velocity and body length. The unsteady Reynolds–Averaged Navier Stokes equations are solved numerically, incorporating an Actuator–Line–modeled propeller. Previously, the authors have shown good comparison to experiment with this approach. The axisymmetric Iowa Body cross–sectional aspect ratio is varied to produce five parametrically–defined hull forms. Flow visualization shows the influence of hull–form geometry on propeller–driven vortex structures. Steady, cross–plane wake profiles emerge by half of a body length downstream. The distribution of momentum and vorticity at this location is uniquely–modified by variations in the upstream geometry. Although propeller swirl is the most significant generator of potential energy, hull effects also alter the thermal–saline distribution, further increasing the total potential energy. Integrated terms from the Vorticity Transport Equation suggest that buoyancy may function as an important driver of vorticity in the developing flow field.
5.1 Introduction

Wakes from propeller–driven bodies transition from a complicated near wake with distinct propeller–blade features to a far wake, in which such features are no longer discernible [12]. The extent of this transition depends on many factors including the geometry and operation of the propulsor [8] as well as the level of flow separation [5]. Some experiments show features of the far wake appearing within twelve propulsor diameters [30], but the development of the far wake can be delayed by appendages on the body [26]. In a stratified flow, swirl from a propeller will mix fluid, generating potential energy in the formation of a mixed patch [7], which can further modify the far wake when the fluid is highly stratified [3, 4, 10, 22, 24]. Variations in the propulsion scheme has a large impact on the strength and structure of the mixed patch [16]. The net–zero–momentum wake was first studied as a disk–with–center–jet (DWCJ) [23] and later as a self–propelled body–of–revolution (BOR) [12, 18, 29]. It is often used to represent a self–propelled vehicle in steady–state operation. In a Computational Fluid Dynamics (CFD) simulation, the propeller may be modeled using fully–resolved rotating geometry or a body–force representation, which includes lifting–line, lifting–surface, and generalized actuator methods [28]. A fully resolved propeller often provides greater fidelity, but its computational requirements are large, making an actuator model a cost–effective alternative [14]. Generalized actuator models are frequently used in the study of wind turbines [19]. The Actuator Line (AL) model imposes an unsteady, rotating body force on the surrounding fluid in a CFD simulation in the place of a propeller using pre–processed lift and drag data [33].

Although non–BOR vehicles have recently received attention [6, 27, 32], the impact of non–BOR geometries on the self–propelled near–wake structure and evolution has not been studied. These asymmetries in hull form, often seen in aquatic life, directly influence the near–field development of the wake and alter the vorticity and potential energy distribution in a stratified environment. Coupled with a propeller, variation in hull form uniquely modifies the near wake fields. The objectives of this study are to:

- Quantify the impact of body shape on the downstream cross–plane profile of velocity, vorticity, and thermal–saline fields.
- Establish the effect of non–BOR hull forms on the evolution of the near wake, with attention to the growth and decay of potential and kinetic energies.
- Identify the relative influence of the source terms of the Vorticity Transport Equation (VTE) in the near–wake generation and transport of vorticity.

The unsteady Reynolds–Averaged Navier–Stokes (RANS) equations are solved numerically to examine the near field of linearly–stratified, turbulent, net–zero–momentum propeller wakes of the Iowa Body [12] modified to provide non–BOR hull forms. The propeller is modeled with the AL method. The authors have previously shown good comparison to experimental
data for the BOR case [15]. Flow visualization is used to show the impact of hull–form variation on vortical structures. Comparison of instantaneous and time–averaged flow fields shows that steady, cross–plane wake profiles appear by half of a body length downstream, suggesting the transition from near to far wake. The hull–form variation directly modifies the velocity and vorticity fields at the inflow of the propeller. This effect is seen in the mixed–patch cross–plane profiles of the flow fields. Because the thermally–stratified environment is isohaline, the mixed–patch temperature–deviation field, in particular, illustrates the structure of the potential energy that is carried downstream into the far wake. Planar integration of potential and kinetic energies show the effect of hull form on the growth and decay of energy. Integrated terms in the Vorticity Transport Equation (VTE) are examined to provide insight into the distribution and evolution of vorticity.

5.2 Approach

5.2.1 Governing Equations

The governing equations of this fluid–flow problem are the unsteady Reynolds–averaged Navier–Stokes equations in Boussinesq form with a body–force term $f_p$ for the unsteady, AL–modeled propeller.

$$\frac{\partial U_i}{\partial x_j} = 0$$

$$\frac{\partial U_i}{\partial t} + \frac{\partial (U_i U_j)}{\partial x_j} = -\frac{1}{\rho_0} \frac{\partial \rho}{\partial x_i} + \nu \frac{\partial^2 U_i}{\partial x_j \partial x_j} + \frac{\partial}{\partial x_j} \overline{u_i' u_j'} + \frac{\Delta \rho}{\rho_0} g_j \delta_{ij} + \frac{1}{\rho_0} f_p$$

In these equations, $U_i$ is the non–inertial velocity, $t$ is time, $\nu$ is the kinematic viscosity, and $g$ is the acceleration due to gravity. Density is expressed as $\rho = \rho_0 + \Delta \rho$, where $\rho_0$ is a reference value and $\Delta \rho$ is the deviation from that value. The gravitational vector $g_j$ points downward in the negative $z$ direction. The upward–positive, vertical position is $z$. This formulation includes the piezometric pressure, $\hat{p} = p - \rho_0 g z$ where $g$ is the magnitude of the gravitational vector.

The governing equations are solved using a custom solver written with the CFD framework OpenFOAM. This custom solver takes into account temperature and salinity transport and the corresponding turbulent fluctuations. The transport of temperature $T$ and salinity $S$ in the stratified environment are determined through the following equations with the diffusion coefficients $\kappa_T$ and $\kappa_S$.

$$\frac{\partial T}{\partial t} + \frac{\partial (U_i T)}{\partial x_j} = \kappa_T \frac{\partial^2 T}{\partial x_j \partial x_j} + \frac{\partial}{\partial x_j} \overline{u_i' T'}$$

$$\frac{\partial S}{\partial t} + \frac{\partial (U_i S)}{\partial x_j} = \kappa_S \frac{\partial^2 S}{\partial x_j \partial x_j} + \frac{\partial}{\partial x_j} \overline{u_i' S'}$$
The Reynolds stresses $\overline{u'_i u'_j}$ and turbulent fluxes $\overline{u'_j t'}$ and $\overline{u'_j s'}$ are determined using a linear eddy–viscosity closure model.

$$
-\overline{u'_i u'_j} = 2\nu_t S_{ij} - \frac{2}{3} k \delta_{ij} \quad (5.5)
$$

$$
-\overline{u'_j t'} = \frac{\nu_t}{\sigma_T} \frac{\partial T}{\partial x_j} \quad (5.6)
$$

$$
-\overline{u'_j s'} = \frac{\nu_t}{\sigma_S} \frac{\partial S}{\partial x_j} \quad (5.7)
$$

Here, $\nu_t$ is the eddy viscosity, $S_{ij}$ is the mean rate of strain, and $k$ is the turbulent kinetic energy. The $k – \omega$ SST turbulence model is selected to compute $\nu_t$ because of its advantage in attached, wall–bounded shear flows [21]. Wall functions are used in the computation of turbulent kinetic energy $k$ and specific turbulence dissipation $\omega$ to relax mesh requirements near the hull. Density is computed using the UNESCO seawater equation–of–state (EOS) [9], but a linear EOS and the TEOS-10 EOS [20] have also been implemented. For this study, the background stratification is isohaline with temperature varying linearly with depth.

### 5.2.2 Numerical Methods

The unsteady mass and momentum equations are solved using the Pressure–Implicit with Splitting of Operators (PISO) method [13]. For each time step, this segregated approach decouples operations on pressure and velocity. First, the momentum equations are solved for velocity using pressure from the previous time step. Then, the pressure–Poisson equation is iteratively solved with corrections to velocity to conserve mass. For these simulations, three iterations are used, each with an additional correction step for mesh non–orthogonality. After the PISO iterations are complete, turbulence quantities are solved for, followed by salinity and temperature. For the towed cases with no propeller, the steady RANS equations are solved using the segregated Semi–Implicit Method for Pressure Linked Equations (SIMPLE) method [25].

The governing equations are discretized in time through implicit, second–order, backward differencing, and in space through the cell–centered finite volume method. A second–order, linear–upwind scheme is applied to the advective term. A first–order upwind scheme is applied to turbulence quantities, and a second–order linear scheme is applied to all other divergence terms. Laplacian terms are discretized using a second–order, linear scheme that is partially–limited to correct for mesh non–orthogonality.

To solve the resulting systems of algebraic equations, two iterative methods are employed. The pressure equation is solved using the Preconditioned Conjugate Gradient (PCG) method with a residual tolerance of $10^{-6}$. The momentum, scalar transport, and turbulence equations are solved using the Pre–Conditioned Bi–Conjugate Gradient (PBiCG) scheme with a residual tolerance of $10^{-8}$. 


5.2.3 Hull Geometry

The hull form of the axisymmetric Iowa Body [12] is varied to consider non–BOR configurations. By default the Iowa Body is a 10.9:1:1 BOR as shown in Figure 5.1, where \( r \) is the radial position, \( x \) is the axial position, and \( L \) is the length of the hull. This study considers non–BOR versions of the Iowa Body, where lateral and vertical dimensions \( D_y \) and \( D_z \) are scaled to give the following five parametrically–defined configurations with aspect ratios: \( D_y/D_z = 2/1, 1.5/1, 1/1, 1/1.5, \) and \( 1/2 \). These configurations are shown in Figure 5.2, where \( y \) is the lateral position, \( z \) is the vertical position, and \( R_0 \) is radius of the BOR Iowa Body. The BOR case is defined by the aspect ratio, \( D_y/D_z = 1/1 \), and the length and displacement volume of the hull are held constant for all geometries. These variations in cross–sectional geometry are representative of a portion of the design space for a self–propelled vehicle.

![Figure 5.1: Standard axisymmetric Iowa Body profile compared to SUBOFF profile.](image)

5.2.4 Actuator Line Model

The unsteady propeller for each non–BOR hull form is simulated using an AL model from the Simulator fOr Wind Farm Applications (SOWFA) library [2]. The AL model considers variations in thickness, twist, pitch, and hydrofoil along the span of each propeller blade by referring to a table of values for the given propeller. Predetermined lift- and drag–coefficient curves determine the appropriate body force to impose on the surrounding fluid at each spanwise station along the propeller blade based on the local flow–angle–of–attack. In this case, the three–bladed Iowa Body propeller is modeled and rotates counter clockwise when viewed from downstream of the hull. The rotation rate is set to provide thrust equal to the drag of the body. Additional details can be found in Jones and Paterson [15].

5.2.5 Computational Mesh

The computational mesh for each hull–form variation was generated using the software cfMesh [1]. Cartesian mesh cells are clustered near the body, propulsor region, and in the wake. The hull is located at a depth of one body length. The inlet, outlet, and far–field
Figure 5.2: Three-dimensional and mid-body, cross-sectional views of Iowa Body hull-form variations.

boundaries are located two body lengths away from the hull. Table 5.1 lists features for the meshes, and Figure 5.3 displays the mesh for the $D_y/D_z = 1/1$ hull-form variation. Because wall functions are used in computing the turbulence variables, the dimensionless wall distance requirement of $y^+ < 100$ was sufficient in capturing boundary layer effects and the viscous drag of the hull. A grid-refinement study of the propeller- and wake-region cells showed that 100 cells/$D_p$ adequately resolved the AL model and downstream wake cross-plane profiles.

Table 5.1: Mesh features.

<table>
<thead>
<tr>
<th>Mesh feature</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total number of cells</td>
<td>$30 \times 10^6$</td>
</tr>
<tr>
<td>Boundary layer cells</td>
<td>$&gt; 20$</td>
</tr>
<tr>
<td>Near-wall mesh spacing</td>
<td>$y^+ &lt; 100$</td>
</tr>
<tr>
<td>Propulsor and wake cells per $D_p$</td>
<td>100</td>
</tr>
<tr>
<td>Maximum non-orthogonality</td>
<td>$&lt; 45^\circ$</td>
</tr>
<tr>
<td>Maximum aspect ratio</td>
<td>$&lt; 170$</td>
</tr>
<tr>
<td>Maximum skewness</td>
<td>$&lt; 0.8$</td>
</tr>
</tbody>
</table>
5.2.6 Initial and Boundary Conditions

Several boundary conditions are employed. Velocity at the inlet is set to the freestream velocity $U_0$ through a Dirichlet boundary condition. The no-slip condition is set on the hull boundary, and the slip condition is set in the far field. The change in pressure in the outlet is set to zero. Background turbulence values of $k$ and $\omega$ are computed assuming a turbulence intensity of 1% and eddy viscosity ratio $\nu_t/\nu$ of 100. Turbulence variables on the hull boundary are computed with wall functions. Other variables satisfy the zero-gradient Neumann boundary condition.

Initial conditions for pressure and velocity are found by solving the potential flow equations. Next, the transient simulation begins using the PISO algorithm. Distributed body forces from the propeller-blades rotate at each time step at the rotation rate of the propeller. The simulation is run until initial-transient flow features advect far downstream and a periodic wake flow is found. For the steady, towed simulations, the AL model is disabled and the SIMPLE algorithm is used.
5.2.7 Flow Conditions

Flow coefficients for this study are selected to represent operation in a typical ocean environment. The Reynolds number is \( Re_L = U_0 L / \nu = 310 \times 10^6 \), where \( U_0 \) is the ship speed. The Froude number \( Fr \) provides a measure of the density stratification, where a small \( Fr \) means strong stratification. To contrast, an infinite \( Fr \) implies zero stratification. A linear temperature stratification with constant salinity is prescribed such that \( Fr = U_0 / ND \approx 350 \) with the Brunt–Väisälä frequency, \( N = (1/2\pi) \sqrt{(-g/\rho_0)(\partial \rho / \partial z)} \). The Richardson number \( Ri \), which is the ratio of buoyancy to flow gradient terms [18], is \( Ri = (g/T_0)(dT/dz)/(dU/dz)^2 \approx 2.54 \times 10^{-3} \). In this equation, \( T_0 \) is a reference temperature, \( \bar{T} \) is the mean temperature, and \( \bar{U} \) is the mean velocity. This small magnitude indicates that the propeller–driven near–wake inertial forces dominate the buoyancy forces. Further downstream in the far field, the local velocity will decay, eventually generating a large \( Ri \) and possible mixed–patch collapse [10].

5.2.8 Energy in the Flow

Downstream kinetic and potential energies in the wake may be studied by integrating these energy fields over axial planes. These energies are studied to help quantify the axial evolution of the wake. The per–unit–length energies \( KE \) and \( PE \) are formulated as follows.

\[
KE_{x,r,\theta} = \int A x, r, \theta \frac{1}{2} \rho U_x^2 + \frac{1}{2} \rho U_r^2 + \frac{1}{2} \rho U_\theta^2 \, dA
\]

\[
PE = \int A \frac{1}{2} \frac{g}{\rho / \partial z} (\rho - \rho_0)^2 \, dA
\]

In these equations, \( A \) is the area covered in a downstream axial plane. For this study the cross–sectional area \( A \) covers a geometric disk of radius \( 15R_p \), where \( R_p \) is the radius of the propeller. The kinetic energy is computed for individual components of velocity in cylindrical coordinates: \( KE_x \), \( KE_r \), and \( KE_\theta \), to assess the respective contributions to the wake energy. By examining the energy of each component, the decay of momentum can be better understood within the context of a spinning propeller wake. The potential energy follows the formulation of Holliday and McIntyre [11].

5.2.9 Vorticity Transport Equation

The Vorticity Transport Equation describes the advection, diffusion, and generation of vorticity throughout the flow. Because of its fundamental significance, the terms of this equation are analyzed. The equation is derived by taking the curl of the Navier–Stokes momentum
Chapter 5. Influence of Hull–Form on the Stratified Near Wake of a Self–Propelled Body

Here, $\omega$ is vorticity and $\tau$ is the viscous stress tensor. The integrated magnitudes of these terms are computed at downstream cross–planes in the wake to compare their relative influence.

5.3 Results

Analysis of the five hull–form variations involves examination of the respective near–wake features within half of a body length downstream, or $1.0 < x/L < 1.5$. Both self–propelled and towed cases are considered. Velocity, vorticity, and temperature fields are the primary flow quantities to be considered. Secondary, integrated quantities further illustrate mechanics of the wake evolution. These integrated quantities include potential energy, kinetic energy, and integrated VTE terms.

5.3.1 Flow Visualization

For a general understanding of the effects of hull–form variation, the flow is visualized using the second invariant of the velocity gradient tensor, $Q$. An iso–surface is taken for each hull–form simulation with $(L/U_0)^2Q = 16.9$ as shown in Figure 5.4. These iso–surfaces and a vertical cutting plane are colored by the velocity defect $U_x/U_0 - 1$, and the visualization extends to half of a body length downstream, where $x/L = 1.5$. Propeller root and tip vortices can be seen trailing off of the stern of the BOR $D_y/D_z = 1/1$ hull form. These vortex structures eventually disappear by $x/L = 1.3$. Variation in hull form modifies the wake, disrupting the helical path of the tip vortices. Modifications to the velocity field result from the non–axisymmetric drag profiles. For the vertically–stretched $D_y/D_z = 1/2$ hull form, a region of decelerated flow is seen on the top and bottom of the wake due to drag. For the laterally–stretched $D_y/D_z = 2/1$, the reverse effect is seen.

5.3.2 Instantaneous and Time–Averaged Fields

Comparison of instantaneous to time–averaged fields reveals the emergence of steady flow quantities. Figure 5.5 compares the instantaneous and time–averaged velocity defect $U_x/U_0 -$
Figure 5.4: Iso–surface of \((L/U_0)^2Q = 16.9\) and vertical cutting plane colored by \(U_x/U_0 - 1\) for each hull form with \(x/L < 1.5\).
Figure 5.5: Comparison of instantaneous to time–averaged $U_x/U_0 - 1$ field for $D_y/D_z = 1/1$ at successive downstream locations.
5.3. Results

1 at various downstream locations for the BOR $D_y/D_z = 1/1$ hull form. Immediately behind the stern of the body, at $x/L = 1.01$, individual propeller–blade wake features are discernible in the instantaneous field in the first column. Predictably, the time–averaged field in the second column shows an axisymmetric profile following the azimuthal path of the rotating propeller. Further downstream at $x/L = 1.10$ individual blade features are still discernible, but by $x/L = 1.45$, the wake is mixed. The axisymmetric profile of the instantaneous field matches that of the time–averaged field. This steady, axisymmetric profile suggests the transition from near– to far–wake regions.

Figure 5.6 shows the velocity defect for the $D_y/D_z = 1/2$ geometry at the same axial locations. Once again, a steady profile emerges by $x/L = 1.45$ after the transient propeller features dissipate. Drag due to the non–BOR hull form redistributes positive and negative momentum. Additional analysis shows that a steady profile emerges by half of a body length downstream for all other non–BOR hull forms.

![Figure 5.6: Comparison of instantaneous to time–averaged $U_x/U_0 - 1$ field for $D_y/D_z = 1/2$ at successive downstream locations.](image)
5.3.3 Cross–Plane Profiles

Cross–plane profiles of primary flow–field quantities are compared at half of a body–length downstream in Figures 5.7 through 5.12. First, the velocity–defect $U_x/U_0 - 1$ of the un–propelled hull forms is shown in Figure 5.7. In the absence of a propeller, the towed profiles are straightforward. The BOR drag wake is axisymmetric, while elongations in the hull form for the non–BOR cases expand the wake extent in the respective $y$ and $z$ directions. With the introduction of a propeller, these flow fields become more complicated.

Figure 5.7: Comparison of $U_x/U_0 - 1$ at $x/L = 1.5$ for each towed hull form.

Figure 5.8: Comparison of $U_x/U_0 - 1$ at $x/L = 1.5$ for each self–propelled hull form.

Figure 5.9: Comparison of $U_\theta/U_0$ at $x/L = 1.5$ for each self–propelled hull form.

Figure 5.8 illustrates $U_x/U_0 - 1$ for the self–propelled hull forms. The BOR wake is axisymmetric with annular thrust and drag regions. The outboard positive momentum from the
5.3. Results

Figure 5.10: Comparison of $\omega_x R_p/U_0$ at $x/L = 1.5$ for each self-propelled hull form.

Figure 5.11: Comparison of $\Delta T/\Delta T_{R_p}$ at $x/L = 1.5$ for each towed hull form.

Figure 5.12: Comparison of $\Delta T/\Delta T_{R_p}$ at $x/L = 1.5$ for each self-propelled hull form.
propeller is counteracted by the inboard negative momentum from the hull. Variation in hull form modifies the distribution of momentum. For the non–BOR $D_y/D_z = 1.5/1$ case, drag from the hull cuts laterally into the annular, thrust region of the propeller, inducing positive momentum regions of increased magnitude. In the $D_y/D_z = 2/1$ case, the same phenomenon is observed, only more pronounced. Regions of drag are intensified laterally and reduced vertically. The $D_y/D_z = 1/1.5$ and $D_y/D_z = 1/2$ cases reveal similar results but rotated 90° counter–clockwise.

Cross–plane profiles of the azimuthal, swirl component of velocity $U_a/U_0$ at $x/L = 1.5$ are presented for each self–propelled hull form in Figure 5.9. The swirl profile of the BOR hull form is axisymmetric, while non–BOR hull forms introduce asymmetries. For $D_y/D_z = 1.5/1$, the profile is elongated diagonally and two regions of increased velocity appear. For $D_y/D_z = 2/1$ this effect is intensified, and two negative–swirl regions appear. Profiles for $D_y/D_z = 1/1.5$ and $D_y/D_z = 1/2$ are similar but rotated 90° counter–clockwise. The positive and negative swirl in these locations suggests regions of circulation, which is confirmed through the axial vorticity field.

The spinning propeller generates axial vorticity in the wake. Cross–plane profiles of the axial vorticity $\omega_x R_p/U_0$ at $x/L = 1.5$ are presented for each self–propelled hull form in Figure 5.10. The BOR hull form results in an axisymmetric profile with positive vorticity in the center and a ring of negative vorticity outward. Variations in hull form warp this feature by generating additional circulations in the two regions of negative swirl. Larger hull–form variations yield larger circulations.

Because these simulations take place in a thermally–stratified environment, any redistribution of the fluid will generate potential energy. For the towed cases, displacement of the fluid from the hull is the sole driver of this redistribution. Temperature perturbations alone can describe the structure of potential energy, because salinity is constant. Figure 5.11 shows the temperature deviation from the background $\Delta T/\Delta T_{R_p}$ for each un–propelled hull form. This temperature deviation is non–dimensionalized by the linear change in temperature over the depth of one propeller radius $\Delta T_{R_p}$. Potential effects from the hull are least–dominant for the BOR $D_y/D_z = 1/1$ case. In the upper semicircle of the wake, warmer fluid from above is entrained, while colder fluid from the interior is expelled. In the lower semicircle, colder fluid from below the wake is entrained and warmer fluid is expelled. This effect is exaggerated for the vertically–stretched hull forms. The largest amount of potential energy is generated by the $D_y/D_z = 1/2$ hull form. For the laterally–stretched non–BOR hull forms, fluid is entrained laterally and expelled vertically, leaving a flow field with relatively colder fluid suspended above warmer fluid. In effect, the sign of the $D_y/D_z = 2/1$ case is opposite to that of the $D_y/D_z = 1/2$ case.

The introduction of the spinning propeller has a large influence on the potential energy. Figure 5.12 shows cross–plane profiles of $\Delta T/\Delta T_{R_p}$ for the self–propelled hull forms at half of body length downstream. For the BOR $D_y/D_z = 1/1$ case, swirl from the propeller drives colder fluid from the bottom of the wake upward and warmer fluid from the top of
the wake downward. The resulting profile is not axisymmetric and shares similarities to profiles of other studies [7]. Variation in hull form further modifies this profile due to potential effects from the hull. Comparison to the towed results of Figure 5.11 helps to visualize this explanation. The towed $D_y/D_z = 2/1$ hull form, for example, transports colder fluid to the upper region of the wake and warmer fluid to the bottom region. Superposing this profile onto the self-propelled $D_y/D_z = 1/1$ case would result in a profile similar to the self-propelled $D_y/D_z = 2/1$ case. Through this exercise, the addition of hull form effects to propeller effects is apparent for all non-BOR variations. To determine differences in the total potential energy of each wake, integrated values must be examined.

5.3.4 Kinetic and Potential Energy

Downstream kinetic and potential energies in the wake are found by integrating these energy fields over axial planes. Kinetic energy is computed individually for the three components of velocity in cylindrical coordinates as described in Equation 5.8. These three kinetic energies are shown in Figure 5.13 for each hull form at half of a body length downstream. Dissipation is driven by viscous and turbulence effects. Radial kinetic energy $KE_r$ decays most rapidly. At the stern location, where $x/L = 1.0$, radial velocity is large due to hull-form potential effects and propulsor forces. Further downstream this velocity component decays as the transition to the far wake occurs. The axial $KE_x$ decays less rapidly than $KE_r$, however the swirl $KE_\theta$ exhibits the smallest decay rate. The relative persistence of swirl velocity is consistent with the observations of Sirviente and Patel [31]. As a consequence of the swirling transport of the fluid, potential energy is generated due to buoyancy forces. For all three velocity components, the widest hull forms maintain the highest levels of kinetic energy, but differences are still small when compared to the BOR $D_y/D_z = 1/1$ hull form.

The evolution of potential energy $PE$ to half of a body length downstream is shown in Figure 5.14 for all hull-form variations. Previous comparison of the cross-plane temperature-deviation profiles in Figures 5.11 and 5.12 illustrated the structure of $PE$, since the environment is isohaline. The coupling of hull-form variation with the propeller yields unique results for each case. This uniqueness is further evident in the $PE$ decay and growth downstream of the body. Immediately behind the stern at $x/L = 1.0$, the $D_y/D_z = 1/2$ hull form generates the largest $PE$. The other hull forms generate less $PE$ than the BOR $D_y/D_z = 1/1$ case at this location. By half of a body length downstream, however, all non-BOR hull forms generate a larger $PE$, with the vertically-stretched hull forms being the highest in magnitude. The potential effects of the non-BOR hull forms contribute to the $PE$, but the spinning propeller is still the dominant source.
5.3.5 Vorticity–Transport Equation Terms

The final analytical method used to investigate the near wake is the comparison of VTE terms from Equation 5.9. The magnitudes of these terms are integrated across axial planes downstream of the BOR $D_y/D_z = 1/1$ hull form and non–dimensionalized by $U_0^2$, as shown in Figure 5.15. The propeller–torque term is not included, because it is zero everywhere except on the propeller plane. By comparing contributions from the respective VTE terms, the generation and transport of vorticity can be better understood. The two most significant contributions come from the unsteady and convective terms. Both of these terms counteract one another, being nearly equal in magnitude and opposite in sign. The next most–significant influence comes from the stretching term with all other terms being much smaller. Such a large discrepancy demonstrates that the inertial forces originally generated by the propeller dominate the wake development to half of a body length downstream. This observation is consistent with the low Richardson and high Froude numbers of the flow. Of the smaller VTE terms, the baroclinic and viscous torques decay most rapidly. The buoyancy term notably does not appear to decay, further–illustrating the relative persistence of $PE$. This buoyancy torque may become an important driver of vorticity in the developing flow field. Integrated VTE terms for non–BOR hull forms are not presented but are similar in magnitude and behavior to those of the BOR hull form. The largest discernible difference among hull forms is found in the buoyancy term, and follows the trends of the $PE$ in Figure 5.14.
5.4 Conclusions

This study investigated the influence of hull–form variation on the stratified near wake of a self–propelled appendageless body, a topic not previously examined. Using an AL model for the propeller, the unsteady RANS equations were solved numerically with the $k – \omega$ turbulence model. Five hull forms of equal displacement volume and length were established with the cross–sectional dimensions: $D_y/D_z = 1/2$, 1.5/1, 1/1, 1.5/1, and 2/1. The permutations in cross–sectional geometry were representative of part of the design space of a self–propelled vehicle. The BOR $D_y/D_z = 1/1$ hull form was compared to the others using several analytical techniques.

Flow visualization illustrated how hull form effects disrupted propeller–driven vortical structures. Comparison of the instantaneous and time–averaged velocity defect field showed that steady cross–plane profiles emerge by half of a body length downstream. Cross–plane profiles at this location revealed the direct influence of each hull form on primary flow–field quantities. Axial velocity defect and temperature deviation appeared to be the superposition of the un–propelled non–BOR profiles with the self–propelled BOR profile, which illustrates the contribution of potential effects of the hull form. Swirl due to the spinning propeller was the primary source of potential energy. Swirl and vorticity profiles also showed the emergence of regions of circulation due to non–BOR geometry. While potential effects due to hull–form
Figure 5.15: Downstream, integrated magnitudes of Vorticity Transport Equation terms for $D_y/D_z = 1/1$. 

CHART: Graph showing integrated magnitudes of Vorticity Transport Equation terms for various terms with different colors and markers, plotted against $x/L$ and $x'/D_p$. The chart indicates the relative contributions of the unsteady, baroclinic, convective, viscous, stretching, and buoyant terms as a function of downstream distance.
variation did influence the structure of the temperature deviation field, propeller effects were dominant. Integrated kinetic and potential energies quantified the small differences among hull form, with the vertically-stretched geometries generating the largest potential energy. Integrated magnitudes of VTE terms were used to examine the underlying generation and transport of vorticity. Analysis of these terms demonstrated the discrepancy between inertial and buoyancy forces as a consequence of the low Richardson and high Froude numbers of the flow. The rapid decay of inertial forces was contrasted, however to the relatively persistent buoyancy forces, which could become important in the generation of vorticity in the developing flow field.
Bibliography


Chapter 6

Deep Learning of the Near Wake with Explicitly–Specified Hull–Form Parameters

Abstract

The contemporary analysis of a self–propelled marine vehicle involves evaluating its near–field wake profile. The large computational expense of accurately simulating the near–field through Computational Fluid Dynamics (CFD) prohibits its use when rapid evaluation is required, such as within a multi–disciplinary design optimization framework. Machine learning is successfully applied to overcome this limitation in expense. A convolutional neural network (CNN) is trained using a database of wake profiles from CFD simulations with variations in the hull–form aspect ratio parameter. Within a fraction of a second, this deep–learning CNN can generate the corresponding wake profile for a given aspect ratio parameter value. A physics–based constraint was implemented to ensure divergence free, incompressible flow in the transverse velocity components, however improvement was negligible, because the generated profiles were already largely divergence–free. The trained network performs well in both inference and prediction and can be readily expanded to include additional geometric parameters.

6.1 Introduction

For the contemporary analysis of a self–propelled marine vehicle, the vehicle can be assessed by examining the flow field near the body. These computations are often too costly when many marine vehicle variants are to be considered. The proposed solution involves leveraging
machine learning. Machine learning networks are often described as “universal approximators” because they have the potential to learn the nonlinear relationships of virtually any problem provided enough depth in the network. Advances in high-performance computing have allowed for larger, more effective machine learning networks. With larger numbers of hidden layers, these “deep”-learning networks have seen widespread success in a broad range of applications including image recognition [11], medical image analysis [23] and even Computational Fluid Dynamics (CFD) [17]. Deep learning has recently been reviewed by several authors [5, 8, 19, 22].

The proposed algorithm is a convolutional neural network (CNN). When trained, this deep-learning tool provides the capability to instantly generate the wake flow field of a given marine vehicle variant based on its cross-sectional aspect ratio \( D_y/D_z \). Figure 6.1 illustrates the CFD “ground truth” training wake profiles associated with the varied hull-form geometries for \( D_y/D_z = 2, 3/2, 1, 2/3, \) and \( 1/2 \). To build the database, the near-field of the self-propelled Iowa Body geometry [12] was previously simulated in a linearly stratified environment with variations in hull-form cross-sectional aspect ratio \( D_y/D_z \) [14]. Each CFD simulation required a moderate computational expense, running for up to 4 days on 512 processors in parallel. Transverse velocity of generated wake profiles is constrained to be divergence free. Although the geometry is only varied through one geometric parameter in this study, the CNN can be easily expanded to incorporate additional parameters. Once trained, the CNN provides a functional mapping between geometry and wake profile that can be evaluated for an arbitrary geometry within a fraction of a second.

![Figure 6.1: Illustration of parameter space for variations in hull form aspect ratio D_y/D_z.](image)

The following section explains the theoretical formulation of the proposed convolutional neural network (CNN). Next, the structure and details of the CNN are presented. The CNN performance in training, inference, and prediction is evaluated. Generated wake profiles for \( D_y/D_z = 1.75 \) and \( D_y/D_z = 2.25 \) are compared to corresponding ground-truth solutions.
that are not included in the training data set. Finally conclusions are discussed.

6.2 Background

6.2.1 Artificial Neural Networks

A brief tutorial on a simple artificial neural network is presented. Figure 6.2 shows a simple fully-connected neural network with 3 inputs, two hidden layers of sizes 4 and 3, and 2 outputs. Fully-connected layers involve multiplying activations $a_i^{(n)}$ by weights $w_{i,j}^{(n)}$ from the preceding nodes, and ultimately from the inputs $Z_i$. In the expression $a_i^{(n)}$, $n$ denotes the layer and $i$ denotes the node in the current layer. The weight $w_{i,j}^{(n)}$ applies to node $i$ in layer $n$ with respect to node $j$ in the previous layer. For example the activation at the top node in the second hidden layer is computed as,

$$a_0^{(2)} = f_{a}^{(2)} \left( w_{0,0}^{(2)} a_0^{(1)} + w_{0,1}^{(2)} a_1^{(1)} + w_{0,2}^{(2)} a_2^{(1)} + w_{0,3}^{(2)} a_3^{(1)} \right)$$  \hspace{1cm} (6.1)

Similarly, the output $\zeta_0$ is computed as,

$$\zeta_0 = f_{a}^{(3)} \left( w_{0,0}^{(3)} a_0^{(2)} + w_{0,1}^{(3)} a_1^{(2)} + w_{0,2}^{(3)} a_2^{(2)} \right)$$  \hspace{1cm} (6.2)

where $f_{a}^{(n)}$ is an activation function, which introduces nonlinearity to the network. Weights in the weight matrix $w$ are learned by minimizing a loss function for the output layer, which compares results to ground-truth data, and back-propagating gradients.

Figure 6.2: Simple, fully-connected neural network with two hidden layers.
Consider a mean–squared–error cost function where \( C \) is the cost or loss of the network.

\[
C = \frac{1}{n} \sum_{i=0}^{n} (\zeta_i - \zeta_{\text{truth},i})^2 = \frac{1}{2}((\zeta_0 - \zeta_{\text{truth},0})^2 + (\zeta_1 - \zeta_{\text{truth},1})^2)
\]  

(6.3)

The loss \( C \) gives a measure of the error in the network by comparing the output values \( \zeta_0 \) and \( \zeta_1 \) to ground–truth values from the training data \( \zeta_{\text{truth},0} \) and \( \zeta_{\text{truth},1} \). A large \( C \) means that the network is performing poorly for the given inputs \( Z_0, Z_1, \) and \( Z_2 \). To train the network to perform better, the weights in the network must be adjusted. These weights are optimized by analytically determining their derivatives with respect to the loss using the chain rule and then back–propagating the loss to incrementally improve each weight. For example, the gradient of the cost function with respect to the weight connecting node 0 in the output layer to node 0 in the previous layer is computed as follows.

\[
\frac{\partial C}{\partial w_{0,0}^{(3)}} = \frac{\partial C}{\partial \zeta_0} \frac{\partial \zeta_0}{\partial w_{0,0}^{(3)}} \quad \text{where}
\]

where

\[
\frac{\partial C}{\partial \zeta_0} = (\zeta_0 - \zeta_{\text{truth},0}) \quad \text{and} \quad \frac{\partial \zeta_0}{\partial w_{0,0}^{(3)}} = f_a^{(3)} \delta_0^{(2)}
\]

(6.4)

(6.5)

Knowing the derivatives of the weights with respect to the loss, the weights are then iteratively adjusted through gradient descent, with \( N_\text{i} \) being the iteration.

\[
w_{0,0}^{(3)N_{\text{i}}+1} = w_{0,0}^{(3)N_{\text{i}}} - \eta_r \frac{C}{\partial C/\partial w_{0,0}^{(3)}}
\]

(6.6)

The learning rate \( \eta_r \) acts as an iterative step size and is commonly set to a small value, such as \( 10^{-3} \) to \( 10^{-5} \) to avoid numerical instability during the optimization process. When large numbers of ground–truth data sets are considered, weights are adjusted by the average \( \partial C/\partial w_{0,0}^{(3)} \) in a process known as stochastic gradient descent.

### 6.2.2 Convolutional Neural Networks

Convolutional neural networks (CNNs) follow similar principles to conventional neural networks, only convolution operations connect layers and the weights of the convolutional kernel are trained. CNNs are well–suited for tasks involving images such as in machine vision, because they have the capacity to recognize spatial features [9]. A brief review of CNNs is presented followed by simple examples of the convolutional operation to demonstrate its usage.

Modern CNNs are sometimes referred to as deep CNNs because of their large numbers of hidden layers. LeCun et al. [18] first applied a deep CNN to the image recognition of handwritten numbers in 1998. Steady improvements in high–performance computing and the
widespread availability of training databases such as ImageNet [4] allowed for the effective use of CNNs in modern computer vision problems. Krizhevsky et al. [16] applied CNNs to the 2010 image recognition competition known as ImageNet, showing significant improvement over the previous state–of–the–art and inspiring many others to follow suite. Their so–called AlexNet was incrementally re–envisioned by others in subsequent years. Simonyan and Zisserman [24] won the 2014 ImageNet competition using a deep CNN with 16–19 layers and small 3 × 3 convolutional kernals. Iandola et al. [13] built the efficient SqueezeNet CNN, which demonstrated similar accuracy to AlexNet but with 50 times fewer parameters. Long et al. [20] successfully applied a fully convolutional network to the problem of image segmentation, in which images are divided into labeled regions of the objects that compose them. Badrinarayanan et al. [2] showed how an encoder–decoder CNN framework could more efficiently solve the same problem. Chen et al. [3] introduced recent advancements to image segmentation, particularly through improvements to upsampling within the network.

To better–understand the network formulation of the present study, the convolution operation is explained through several graphical examples. These visualizations are inspired by those of Dumoulin and Visin [6]. The convolution operation can be described by four attributes: kernal dimensions, stride, input dimensions, and padding. A simple convolution example is shown in 6.3 with the following attributes.

- 3 × 3 kernal
- 1 × 1 stride
- 4 × 4 input
- 1 × 1 zero padding

The 4 × 4 input layer is shown in orange in the background with single–celled zero padding shown as clear, outlined cells. The zero padding cells contain an activation value of zero. The 3 × 3 kernal, which is highlighted in gray, slides along the input layer moving one cell at a time as specified by the 1 × 1 stride. The kernal is composed of 9 weights. As the kernal incrementally moves along the input layer, the inner product is taken between the kernal weight matrix and activations of the highlighted cells. The resultant value is than set to the output cell highlighted in gray in the foreground, output layer. By selecting 1 × 1 zero padding and 1 × 1 stride, the spatial size of the output layer remains the same as the input layer.

A second convolution example is shown in Figure 6.4 with the following attributes.

- 4 × 4 kernal
- 2 × 2 stride
- 6 × 6 input
Chapter 6. Deep Learning of the Near Wake with Explicitly-Specified Hull-Form Parameters

Figure 6.3: $3 \times 3$ kernal over $4 \times 4$ input with $1 \times 1$ border of zeros and $1 \times 1$ stride.

- $1 \times 1$ zero padding

In this case the stride is $2 \times 2$ so the convolutional kernal moves two cells at a time over the $6 \times 6$ input. By increasing the stride, this convolution operation decreases the spatial size of the output layer to half of that of the input layer. In this case the spatial size is reduced from $6 \times 6$ in the input layer to $3 \times 3$ in the output layer. It is also possible to increase the spatial size between layers.

Figure 6.4: $4 \times 4$ kernal over $6 \times 6$ input with $1 \times 1$ border of zeros and $2 \times 2$ stride.

To increase spatial size from layer to layer, fractionally-strided convolutions are employed. These operations are sometimes referred to as deconvolutions, which is not technically correct, or transposed convolutions. An example of a fractionally-strided convolution is shown in Figure 6.5. This operation is the transpose of the previous convolution with a $4 \times 4$ kernal over a $6 \times 6$ input with $1 \times 1$ padding and a stride of $2 \times 2$. It is equivalent to a convolution with the following properties.

- $4 \times 4$ kernal
• 1 × 1 stride
• 3 × 3 input with zeros in–between
• 2 × 2 zero padding

In this example the spatial size is doubled from 3 × 3 to 6 × 6.

Figure 6.5: Transpose of 4 × 4 kernal over a 6 × 6 input with 1 × 1 padding and a stride of 2 × 2, which is equivalent to a convolution with a 4 × 4 kernal over a 3 × 3 input with 2 × 2 zero padding and a stride of 1 × 1.

These convolutional operations fundamentally connect layers in a CNN. In a typical CNN, layers have more than one feature map. The number of feature maps is referred to as the number of channels in the layer. If there are multiple feature maps from the input layer, then a separate convolution kernal operates on each layer and the results are summed for the output feature map. Each output feature map will be a function of every input feature map. As an example, if an input layer has 4 channels \( C_i = 4 \), 2 output channels \( C_o = 2 \), and a 3 × 3 convolutional kernal, the number of learnable kernal weights are \( 3 \times 3 \times 4 \times 2 = 72 \). The structure of the CNN employed in this study will be discussed.

6.3 Approach

6.3.1 Network Structure

At a high level, the CNN in this study can be described as a function that takes in hull–form geometric parameters as the input array \( Z_p = [z_1, z_2, z_3, ... z_p] \) and outputs the corresponding wake profile.

\[
\text{wake profile} = f(Z_p)
\]  

(6.7)

For this study, only the hull form aspect ratio parameter is considered, so \( Z_p = [D_y/D_z] \).
The CNN used in this study is visualized in Figure 6.6. A sequence of fractionally–strided convolutions project and reshape the input parameter array $Z_p$, into the flow–field velocity components on a $64 \times 64$ cell data plane. Each block is described by a width and height in the lateral directions, and a number of channels in the depth–wise direction. For example, the first block has a spatial width of $4 \times 4$ and 256 channels. Layers with narrower spatial dimensions have a larger number of channels to facilitate learning. Convolutional weights are learned iteratively through the ADAM optimization scheme [15]. The weights are initialized through Xavier initialization [7]. The network has a total of 702,439 learnable parameters and a state size of 73,729. It was built in and trained using PyTorch [1].

From the input layer to the output layer, the CNN takes the following operations, where “Conv” corresponds to the convolution operation.

- **Conv 1**: fractionally–strided, $4 \times 4$ kernal, $0 \times 0$ zero–padding, $1 \times 1$ strides, PReLU activation. Instance normalization is applied between the first and second layers to improve training [26].

- **Conv 2-4**: fractionally–strided, $4 \times 4$ kernal, $1 \times 1$ zero–padding, $2 \times 2$ strides, PReLU activation.

- **Conv 5**: fractionally–strided, $4 \times 4$ kernal, $1 \times 1$ zero–padding, $2 \times 2$ strides, tanh activation.

Dropout regularization [25] was tested but did not improve results so was not included in the final network. Training data for each flow field is pre–processed by normalizing by the maximum magnitude. This ensures that all flow–field data varies between -1 and 1 in the network, which is within the bounds of the tanh activation function in the final layer. The reverse normalization operation is applied to output wake profiles. The Parametric Rectified Linear Unit (PReLU) activation function [10] is used for all activations except for the final
layer, which uses the hyperbolic tangent function tanh. For an input $x_I$, the PReLU is function is,

$$\text{PReLU}(x_I) = \begin{cases} 
    x_I & x_I \geq 0 \\
    ax & x_I < 0 
\end{cases}$$

(6.8)

with a learnable slope parameter $a$. The PReLU function was chosen over the conventional Rectified Linear Unit (ReLU) [21], because it effectively avoided dead activations that appeared when using ReLUs. The sloped curve described by $a$ allows the network to back–propagate gradients even when negative inputs are given to the PReLU. Figure 6.7 plots both the standard ReLU and PReLU activation functions. Nonlinearity is present in the function due to the discontinuity at 0.

![Figure 6.7: Comparison of ReLU and PReLU activation functions.](image)

Hyperparameters are listed in Table 6.1. The learning rate is initially set to $\eta_r \equiv 10^{-4}$ for the first 10,000 epochs. It is then set to $\eta_r \equiv 10^{-5}$ until epoch 100,000, and finally $\eta_r \equiv 10^{-6}$ for epochs beyond 100,000. Incrementally reducing the learning rate reduces numerical instabilities in the learning process. The $\beta_1$ and $\beta_2$ parameters of the ADAM optimization scheme are set to 0.5 and 0.999 respectively.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Learning rate $\eta_r$</td>
<td>$10^{-4}, 10^{-5}, 10^{-6}$</td>
</tr>
<tr>
<td>ADAM $\beta_1, \beta_2$</td>
<td>0.5, 0.999</td>
</tr>
</tbody>
</table>

### 6.3.2 Physics–Based Constraint: Divergence Free Flow

A physics–based constraint was implemented in an attempt to enforce more accurate solutions. However, generated profiles without the constraint were already largely divergence free, so the constraint did not noticeably improve results. The constraint was also limited
Chapter 6. Deep Learning of the Near Wake with Explicitly-Specified Hull-Form Parameters

146

...to transverse components of velocity and not the axial component. As such, this constraint is only briefly discussed.

The low-Reynolds number flow of the marine vehicle considered in the training set was incompressible. CFD solutions were divergence free to satisfy this condition. The incompressibility condition is,

$$\nabla \cdot \vec{U} = 0$$ (6.9)

where \( \vec{U} \) is the 3-component velocity vector \( U_x \hat{i} + U_y \hat{j} + U_z \hat{k} \). Downstream of the body, axial gradients are small, and the cross-plane wake profile only covers the transverse, \( y-z \) plane, so only \( y \) and \( z \) components of velocity can be considered,

$$\frac{\partial U_x}{\partial x} + \frac{\partial U_y}{\partial y} + \frac{\partial U_z}{\partial z} = 0$$ (6.10)

The consequence of this simplification is that the divergence-free condition can only constrain transverse components of velocity. Any nonzero \( \partial U_y/\partial y + \partial U_z/\partial z \) can be considered an error in the generated wake profiles.

During network training this constraint was implemented at each epoch. First the network weights were adjusted by comparing the generated wake profiles to those of the training cases. Error was backpropagated using conventional stochastic gradient descent. Afterwards, a set of 5 wake profiles was generated from randomized inputs of \( D_y/D_z \). For each generated profile, a divergence-free velocity field was explicitly calculated using the finite difference method. The divergence-free field was computed first using the generated \( U_y \) in its boundary conditions, followed by stochastic gradient descent. The field was then computed using the generated \( U_z \) in its boundary conditions, followed by stochastic gradient descent. The overall effect was a reduction in the \( \partial U_y/\partial y + \partial U_z/\partial z \) error term, however \( U_y \) and \( U_z \) flow fields were visibly unaffected because \( \partial U_y/\partial y + \partial U_z/\partial z \) was already small even when the network was trained without this constraint.

6.4 Results

6.4.1 Network Performance

The network performance is evaluated. Even though the CNN is trained to generate profiles for multiple flow variables including all components of velocity, only the axial component of the velocity defect \( U_x/U_0 - 1 \) is presented here for brevity. Transverse velocity components \( U_y/U_0 \) and \( U_z/U_0 \), which perform similarly, are not shown. The iterative training process is explained first.

Figure 8.11 shows the iterative training of the network. Training error is computed as the mean-squared-error (MSE) of generated wake profiles with respect to the CFD training
data set. Test error is the MSE of generated wake profiles when compared to a test case not included in the training data. Both errors are non-dimensionalized by the training MSE at the first epoch. The figure shows that both training and test error decrease dramatically then level off. The test error levels off relatively quickly, which shows that network performance is quickly established and improves little after that point. Additionally, the test error does not increase, which demonstrates that the network is not over–fitting the training data. The total training time was approximately 8 days using 1 Intel Skylake Xeon Gold 3 Ghz CPU. Graphical Processing Unit (GPU) acceleration was not employed in this training. Once trained the network can be evaluated in approximately 0.2 seconds.

![Figure 6.8: Training and test error.](image)

After training, the network can generate wake profiles of the velocity flow field given an arbitrary \( D_y/D_z \) hull–form input. Here, the network is shown to reproduce the training data. Figure 6.9 shows generated wake profiles, ground–truth wake profiles, and the associated error between the two. Columns correspond to the 5 separate hull–form aspect ratios. The top row shows wake profiles generated from the CNN, the second row shows wake profiles taken from the CFD database, and the third row shows the error. MSE is quantified at the top of each error plot as the average MSE over the plane. Flow field contours are colored by the non–dimensional velocity \( U_x/U_0 - 1 \) of the flow field. As shown, the generated wake profiles closely match the ground truth CFD training data.

First, the CNN performance is examined in inference. Figure 6.10 shows the CNN–generated wake profile for \( D_y/D_z = 1.75 \) compared against the ground–truth CFD profile for \( D_y/D_z = 1.75 \), which was excluded from the training data set. The first plot shows the generated flow field, the second shows the CFD solution, and the thirst shows the error with the average MSE listed above the plot. A visual inspection shows that the CNN does a good job of approximating the profile for \( D_y/D_z = 1.75 \). Additionally the quantified MSE is small.
Figure 6.9: Network performance in training.

Figure 6.10: Network performance in inference for $D_y/D_z = 1.75$. 
6.4. Results

Next, CNN network performance in prediction is examined. Figure 6.11 shows the CNN–generated wake profile for $D_y/D_z = 2.25$ compared against the ground–truth wake profile for the same $D_y/D_z$, which was excluded from the training data. Error is larger for this case than for the inference case. Additionally, anomalous distortions can be seen outside of the turbulent wake region, however they are small in magnitude. Overall, the network does a good job of capturing the important wake features.

![Figure 6.11: Network performance in prediction for $D_y/D_z = 2.25$.](image)

The CNN performance is also compared against conventional interpolation and extrapolation. Figure 6.12 compares network performance in inference to linear interpolation. The CNN–generated wake profile for $D_y/D_z = 1.75$ is compared against a profile computed from linear interpolation of the $D_y/D_z = 1.5$ and $D_y/D_z = 2.0$ training data profiles. The ground–truth $D_y/D_z = 1.75$ profile is also shown and MSE is listed in the plot titles. Both the machine–learning and interpolated profiles show similar structure and comparable error, however the error of the machine–learning algorithm is smaller.

![Figure 6.12: Network performance compared to linear interpolation for $D_y/D_z = 1.75$.](image)

Figure 6.13 compares network performance in prediction to linear extrapolation. The CNN–
Chapter 6. Deep Learning of the Near Wake with Explicitly-Specified Hull-Form Parameters

6.5 Conclusions

A deep-learning convolutional neural network was constructed and trained to learn the functional relationship between the hull-form cross-sectional aspect ratio of self-propelled vehicle and its wake profile. Convolutional operations were employed in the network because they have the capacity to recognize visual features in images. Evaluation of the CNN demonstrated good performance in training, inference, and prediction, showing smaller error than conventional linear interpolation techniques. In the case of prediction, anomalous regions were present, however their magnitudes were small. A physics-based constraint for divergence free flow had a negligible effect on network performance, because only transverse components of velocity could be considered and the generated wake profiles were largely divergence free even without the constraint. The CNN can be readily expanded to include additional geometric parameter inputs as well as a broader training database to learn from.
Bibliography


Chapter 7

Parameterized Hull Form: Influence of Cross–Section, Sail Appendage, and Slenderness

Abstract

Geometric features of the hull from of a self–propelled marine vehicle will influence the surrounding flow field. However, the exact relationship between the overall shape and the near–field wake evolution is not well understood. Characterizing the sensitivities to changes in specific geometric features will help to understand how hull–form geometric parameters can impact the flow field in the surrounding ocean medium. In this study, these sensitivities are investigated. The unsteady Reynolds–Averaged Navier Stokes (RANS) equations are solved for a self–propelled marine vehicle in a linearly–stratified environment with variations in cross–sectional aspect ratio, sail aspect ratio, and length–to–diameter ratio. In total 40 RANS simulations were conducted to capture the effects of these variations. The parameterized hull form is formulated to represent a generic marine vehicle with a sail appendage and three sections: a parabolic forebody, a parallel midbody, and an elliptic afterbody. Parameterization of this geometry allows for wide range of hull forms to be investigated, more so than would be available for other generic geometries already studied in the literature. Numerical methods were validated against experimental data for the Iowa Body geometry in a previous study. The INSEAN E1619 propeller used in these self–propelled simulations through the actuator–line method is compared to open–water experimental data. Simulations are conducted at Reynolds numbers of $Re_L = 400 \times 10^6$ and $Re_L = 280 \times 10^6$ based on the hull length and freestream velocity. Near–field analysis shows the impact of the sail appendage on wake evolution. Downstream cross–plane wake profiles illustrate sensitivities to variations in hull geometry. Integrated potential and kinetic energies are computed for each profile.
7.1 Introduction

The flow field around a self-propelled vehicle is characterized by complicated interactions between the hull and propulsor on the surrounding environment. In operational conditions, turbulence encompasses the evolution of the wake. Originally studied as a disk–with–center–jet (DWCJ) \[41\], and later with more realistic geometries \[16, 24, 25, 36, 54, 62\], experiments have helped to better-understand turbulent, propeller-driven, net-zero-momentum (NZM) wakes. The discrete vortex structures formed immediately behind the propeller blades eventually interact and break down until a nearly-axisymmetric wake is formed \[26\]. Sirviente and Patel \[57\] have shown that this type of transition can occur within approximately twelve initial propulsor diameters, however simulation shows that stern appendages can delay the transition \[48\]. Swirl in the wake persists longer than the axial velocity \[58\]. Reynolds number, propeller geometry, and propeller operating conditions have been shown to effect the formation, evolution, and breakdown of vortex structures in the wake \[18\]. The fundamentals of vortex breakdown are still an extensive area of research \[38\].

The presence of stratification in the surrounding environment adds another layer of complexity. Swirl from the propulsor, turbulent mixing, and potential effects from the vehicle will result in the formation of a mixed patch within the wake \[15\]. In the far field this mixed patch can collapse when buoyancy forces become dominant \[7, 12, 22, 40\]. A common approach to simulating the far field is to solve the parabolized Navier Stokes equations over a 2D computational mesh in time. The 2D wake profile that initializes these simulations can be referred to as an initial data plane (IDP). Assuming axial gradients are small, these “2D+t” simulations provide the evolution of a wake over large scales at a small computational cost. Numerous experiments have been conducted to investigate the stratified wake evolution, generation of internal gravity waves, and coherent structures \[37, 60, 61\]. Direct Numerical Simulation (DNS) has been used to study the fundamental physics of the stratified wakes of towed spheres \[20, 28, 45, 46, 47\]. Redford et al. \[51, 52\] investigated the wake energy budget within the three wake regimes: 3D near field, non-equilibrium wake collapse, and quasi-two-dimensional (Q2D) far field. Brucker and Sarkar \[6\] compared towed and self-propelled wakes in a stratified fluid, showing that the self-propelled wake realizes the same three regimes although decays more rapidly than in the towed case. Buoyancy was shown to decouple regions of positive and negative momentum in the self-propelled case allowing those regions to decay at different rates. Excess momentum in the self-propelled case increases turbulent kinetic energy and leads to qualitative changes in the downstream vortical structures \[13\]. Reduction in potential energy in the near-field thermal-haline distribution will directly influence the strength of far-field buoyancy effects.

Several methods exist for modeling propellers. Fully resolving the propeller geometry within a Computational Fluid Dynamics (CFD) simulation provides the highest fidelity at the highest expense. Cheaper but lower fidelity methods include panel/lattice methods and actuator models \[53\]. Of the generalized actuator models, the actuator line (AL) model is capable of simulating unsteady effects by projecting a distribution of body forces along the
span of each propeller blade within a CFD simulation. This method provides a cost-efficient alternative to a fully-resolved propeller with moderate agreement to experimental data [29].

Turbulence modeling presents a major challenge in CFD. While DNS can resolve turbulence down to the smallest scales, it is prohibitively expensive for realistic Reynolds numbers. Solving the Reynolds-Averaged Navier Stokes (RANS) equations is cheaper but less accurate than Large Eddy Simulation (LES) or hybrid RANS/LES methods due to large modeling uncertainties [64]. In 2014, Slotnick et al. [59] predicted the future use of turbulence modeling techniques in CFD until 2030, finding that RANS would continue to see widespread use, however LES-based techniques would see increasing use as computational resources improved.

Bensow [5] surveyed CFD methods in ship hydrodynamics finding that while potential flow methods are computationally efficient, the present state-of-the-art requires the more advanced RANS, LES, and hybrid RANS/LES methods. Chase and Carrica [9, 10] simulated the open-water performance of the INSEAN E1619 propeller using RANS, Delayed Eddy Simulation (DES), and Delayed Detached Eddy Simulation (DDES) and comparing the results to experimental measurements. The hybrid RANS/LES methods performed best and the RANS method was overly dissipative. RANS can still provide solutions with moderate accuracy, however. In 2018, Wang et al. [63] used RANS to investigate open-water propeller skew finding that increasing propeller skew modified the interactions between vortex structures thereby delaying wake breakdown.

The INSEAN E1619 seven-bladed propeller serves as generic marine vehicle propeller commonly referenced in the literature. Its appearances in published experimental and computational studies is shown in Table 7.1. The authors, year, configuration, use of Experimental Fluid Dynamics (EFD), and use of Computational Fluid Dynamics (CFD) is tabulated. The propeller is commonly simulated in open water to compare to experimental measurements. It is also popularly used as the propulsor for the Defense Advanced Research Projects Agency (DARPA) SUBOFF generic submarine geometry. Original experiments for the propeller were performed in 2009 by Di Felice et al. [14] and Liefvendahl et al. [35] using Laser Doppler Velocimetry (LDV) to half of a diameter downstream. Only in 2018 have additional experiments been made. The propeller is shown in Figure 7.3. In order to simulate the propeller using the actuator-line method, spanwise characteristics of chord, thickness, twist, and skew were extracted from the geometry. These characteristics are shown in Figure 7.4.

The present work reveals modifications to the near-field of a self-propelled generic hull form due to changes in geometry. The introduction of a sail appendage results in a localized deficit of axial momentum and small increase in potential energy in the cross-plane wake profile. Asymmetric hull forms similarly redistribute axial momentum also modifying the potential energy. Effects of a slenderness are small. Parametric changes in geometry modify the distribution of other flow fields to a lesser degree.

In this study, the unsteady RANS equations are solved to examine the self-propelled, stratified near-wake of a generic hull form. The parameterized geometry is varied in cross-sectional aspect ratio, sail aspect ratio, and length-to-diameter ratio. Near-field open water
Table 7.1: Marine-vehicle, turbulent wake experiments.

<table>
<thead>
<tr>
<th>Author</th>
<th>Year</th>
<th>Configuration</th>
<th>EFD</th>
<th>CFD</th>
</tr>
</thead>
<tbody>
<tr>
<td>Di Felice et al. [14]</td>
<td>2009</td>
<td>open water</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Liefvendahl et al. [32, 33, 35]</td>
<td>2010</td>
<td>open water, AFF8 submarine, and DTMB 5415 surface combatant</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Liefvendahl &amp; Troëng [34]</td>
<td>2011</td>
<td>SUBOFF</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Schroeder &amp; Balaras [56]</td>
<td>2011</td>
<td>open water</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>Liefvendahl &amp; Troëng [31]</td>
<td>2012</td>
<td>SUBOFF</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Chase et al. [10, 11]</td>
<td>2012</td>
<td>open water &amp; SUBOFF</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Schroeder [55]</td>
<td>2014</td>
<td>open water &amp; SUBOFF</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Balaras, Schroeder, &amp; Posa</td>
<td>2015</td>
<td>open water &amp; upstream appendage</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Posa &amp; Balaras [50]</td>
<td>2016</td>
<td>open water &amp; SUBOFF</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Özden et al. [42]</td>
<td>2016</td>
<td>open water &amp; SUBOFF</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Özden and Çelik [44]</td>
<td>2017</td>
<td>open water &amp; SUBOFF</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Burunsuz et al. [8]</td>
<td>2017</td>
<td>open water &amp; SUBOFF</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Liang et al. [30]</td>
<td>2017</td>
<td>SUBOFF</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Felli and Falchi [17]</td>
<td>2018</td>
<td>open water</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Özden et al. [43]</td>
<td>2018</td>
<td>open water &amp; SUBOFF</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Posa et al. [49]</td>
<td>2018</td>
<td>open water</td>
<td>X</td>
<td>X</td>
</tr>
</tbody>
</table>

Simulations of the INSEAN E1619 propeller are compared to experimental measurements to demonstrate the capabilities of the AL-modeled propeller used in these simulations. A grid study for the case of an axisymmetric hull with a sail shows numerical convergence as well as the near-field vortex structure and wake evolution. Downstream wake profiles are evaluated at the IDP, which is specified at 5 body diameters or 10 propeller diameters downstream. A survey of geometries are considered based on permutations in geometric parameters. Cross-sectional aspect ratio, sail aspect ratio, and length-to-diameter ratio are all varied. Integrated potential and kinetic energies are computed for each case.

7.2 Approach

7.2.1 Governing Equations

The governing equations for this fluid-flow problem are the unsteady Reynolds-averaged Navier-Stokes (RANS) equations in Boussinesq form. The body force term $f_p$ is included
for the actuator–line (AL) propeller model.

\[
\frac{\partial U_i}{\partial t} + \frac{\partial (U_i U_j)}{\partial x_j} = -\frac{1}{\rho_0} \frac{\partial \rho}{\partial x_i} + \nu \frac{\partial^2 U_i}{\partial x_j \partial x_j} + \frac{\partial}{\partial x_j} \overline{u_i' u_j'} + \frac{\Delta \rho}{\rho_0} g_j \delta_{ij} + \frac{1}{\rho_0} f_p
\]

Terms in the equations include: non–inertial velocity \( U_i \), time \( t \), kinematic viscosity \( \nu \), and density \( \rho \). The density is written in terms of a background and a perturbation component: \( \rho = \rho_0 + \Delta \rho \). The vertical coordinate is \( z \), and the gravitational vector \( g_j \) points in the negative \( z \) direction. The piezometric pressure is \( \hat{p} = p - \rho_0 g z \), where \( g \) is the magnitude of \( g_j \).

A custom solver was written using the OpenFOAM CFD framework to solve the governing equations. Salinity \( S \) and temperature \( T \) transport equations are solved, along with the corresponding turbulent fluctuations.

\[
\frac{\partial T}{\partial t} + \frac{\partial (U_j T)}{\partial x_j} = \kappa_T \frac{\partial^2 T}{\partial x_j \partial x_j} + \frac{\partial}{\partial x_j} \overline{u_j' u_j'}
\]

Here, \( \kappa_T \) and \( \kappa_S \) are diffusion coefficients. A linear eddy–viscosity closure model is used to compute the Reynolds stresses \( \overline{u_i' u_j'} \) and turbulent fluxes \( \overline{u_j' T} \) and \( \overline{u_j' S'} \).

\[
\begin{align*}
-\overline{u_i' u_j'} &= 2 \nu_t S_{ij} - \frac{2}{3} k \delta_{ij} \\
-\overline{u_j' T} &= \frac{\nu_t}{\sigma_T} \frac{\partial T}{\partial x_j} \\
-\overline{u_j' S'} &= \frac{\nu_t}{\sigma_S} \frac{\partial S}{\partial x_j}
\end{align*}
\]

In these equations, \( \nu_t \) is the eddy viscosity, \( S_{ij} \) is the mean rate of strain, and \( k \) is the turbulent kinetic energy. The \( k - \omega \) SST two–equation turbulence model is selected to compute \( \nu_t \) because of its advantage in computing the attached flows over a bodies [39]. Wall functions are used to solve for \( k \) and \( \omega \), the specific turbulence dissipation. Density is computed through the UNESCO seawater equation of state [19].

### 7.2.2 Numerical Methods

The Navier–Stokes governing equations are solved using a custom OpenFOAM solver [3] through the Pressure–Implicit with Splitting of Operators (PISO) method [27]. The PISO method decouples operations on the velocity and pressure variables. The following procedure is followed at every time step.
1. The momentum equations are solved for velocity using pressure from the previous time step.

2. The pressure–Poisson equation is solved iteratively with corrections to velocity to conserve mass. Three inner iterations are used, each including a mesh non–orthogonality corrector step.

3. Turbulence quantities are solved for using the $k - \omega$ SST two–equation turbulence model.

4. Salinity and temperature are solved for.

5. The time step is advanced.

Implicit, second–order, backward differencing is used in temporal discretization, while the cell–centered finite volume method is used in spatial discretization. The second–order, linear–upwind scheme is applied to the advective term of the momentum equations. The first–order, upwind scheme is applied to turbulence quantities. The second–order, linear scheme is applied to all other divergence terms. Laplacian terms are discretized using a second–order, linear scheme that is partially–limited to correct for mesh non–orthogonality.

The resulting algebraic equations are solved using two iterative methods. The Preconditioned Conjugate Gradient (PCG) method is used to solve the pressure equation with a residual tolerance of $10^{-6}$. The Pre–Conditioned Bi–Conjugate Gradient (PBiCG) scheme is used to solve the momentum, scalar transport, and turbulence equations with a residual tolerance of $10^{-8}$.

### 7.2.3 Actuator–Line Model

The unsteady INSEAN E1619 propeller is modeled using the AL method from the Simulator fOr Wind Farm Applications (SOWFA) library [2]. In this approach a distributed of force $f_p$ is projected onto discrete mesh cell volumes in place of physically–resolved propeller blades.

$$f_p(r) = \frac{F_p}{\varepsilon^3 \pi^{3/2}} \exp \left[ -\left( \frac{r}{\varepsilon} \right)^2 \right]$$  \hspace{1cm} (7.8)

Here, $F_p$ is the discrete actuator element force composed of lift $F_L$ and drag $F_D$. The variable $r$ is the distance between CFD cell center and actuator point, and $\varepsilon$ controls the Gaussian width of the projected force. Lift and drag are computed as the following.

$$F_L = \frac{1}{2} C_l(\alpha) \rho U_{rel}^2 c w, \quad F_D = \frac{1}{2} C_d(\alpha) \rho U_{rel}^2 c w$$  \hspace{1cm} (7.9)

Here $\rho$ is the density, $U_{rel}$ is local flow speed, and $\alpha$ is the local flow angle–of–attack. The actuator–sectional chord and width are $c$ and $w$, respectively. The functional relationships
7.2. Approach

for lift and drag coefficients, \( C_f(\alpha) \) and \( C_d(\alpha) \), are determined using lookup tables, so the user must provide accurate lift and drag data for each hydrofoil section. Lift and drag terms can be determined through experiment, additional simulation, or analytical techniques.

### 7.2.4 Parameterized Generic Hull Geometry

A parameterized, generic hull geometry is considered. The geometry of overall length \( L \) is composed of three sections: a parabolic forebody, a parallel midbody, and an elliptic afterbody. The lengths of these sections are \( L_f, L_{PMB}, \) and \( L_a \), respectively, so \( L = L_f + L_{PMB} + L_a \). The hull has a nominal maximum diameter of \( D \) in the body of revolution (BOR) case. For non–BOR cases, \( D_y \) and \( D_z \) are the maximum horizontal and vertical dimensions of the hull. A virtual \( D \) can be computed as \( D = D_y/(D_y/D_z)^{1/2} = D_z/(D_y/D_z)^{-1/2} \), such that BOR and non–BOR cross–sectional areas are equivalent. Assigned geometric ratios for the generic hull form are listed in Table 7.2.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( L_f/D )</td>
<td>2.0</td>
</tr>
<tr>
<td>( L_a/D )</td>
<td>4.0</td>
</tr>
<tr>
<td>( L/D )</td>
<td>variable: 7.0 – 11.0</td>
</tr>
<tr>
<td>( L_{PMB}/D )</td>
<td>variable: ( (L - L_f - L_a)/D )</td>
</tr>
</tbody>
</table>

The equations describing the three hull sections are presented.

\[
\begin{align*}
  r_f(x) &= \frac{D}{2} \left( 1 - \left( \frac{L_f - x}{L_f} \right)^{n_{f1}} \right)^{1/n_{f2}} \quad (7.10) \\
  r_{PMB}(x) &= \frac{D}{2} \quad (7.11) \\
  r_a(x) &= \frac{D}{2} \left( 1 - \left( \frac{x - (L_f + L_{PMB})}{L_a} \right)^{n_{a1}} \right)^{1/n_{a2}} \quad (7.12) \\
  r_{hull}(x) &= \begin{cases} 
  r_f(x) & 0 \leq x \leq L_f \\
  r_{PMB}(x) & L_f < x \leq L_f + L_{PMB} \\
  r_a(x) & L_f + L_{PMB} < x \leq L 
\end{cases} \quad (7.13)
\end{align*}
\]

In the preceding equations, \( x \) is the axial station along the hull where \( x = 0 \) is at the bow and \( x = L \) is at the stern, \( r_{hull}(x) \) describes the nominal radial position, which varies in form depending on the section of the hull. The exponents \( n_{f1}, n_{f2}, n_{a1}, \) and \( n_{a2} \) are shape factors controlling the fullness and bluntness of the hull. For example, \( n_{f1} = n_{f2} = 1 \) gives a conical form, whereas \( n_{f1} = n_{f2} = 2 \) gives an elliptic form. Shape factors for the generic hull form...
are presented in Table 7.3. Figure 7.1 compares the generic hull form BOR profile to two other commonly-used geometries in the literature: the Iowa Body [26] and SUBOFF [21].

Table 7.3: Shape factors assigned to the generic hull form.

<table>
<thead>
<tr>
<th>Shape factor</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$n_{f1}$</td>
<td>2.0</td>
</tr>
<tr>
<td>$n_{f2}$</td>
<td>2.4</td>
</tr>
<tr>
<td>$n_{a1}$</td>
<td>2.3</td>
</tr>
<tr>
<td>$n_{a2}$</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Figure 7.1: Generic hull form BOR profile with $L/D = 10$ compared to the Iowa Body and SUBOFF.

For non-BOR cases, $D_y$ is the maximum horizontal dimension and $D_z$ is the maximum vertical dimension excluding any appendages. Horizontal and vertical positions, $y_{hull}$ and $z_{hull}$, are found through the following equations, where $\theta$ is the azimuthal angle.

\[
y_{hull}(x, \theta) = \left(\frac{D_y}{D_z}\right)^{1/2} r_{hull}(x) \cos \theta
\]

\[
z_{hull}(x, \theta) = \left(\frac{D_y}{D_z}\right)^{-1/2} r_{hull}(x) \sin \theta
\]

These equations consider non-BOR forms by taking into account the hull-form aspect ratio $D_y/D_z$.

The generic hull form also considers a sail appendage. This simple appendage is an extruded NACA 0020 hydrofoil with a cap of thickness $t_{cap}$ that spans to $1/4t$ where $t$ is the chord-wise local thickness of the hydrofoil. The sail is described by chord length $c$, aspect ratio $b/c$, and leading-edge (LE) axial location $x_{LE}$. Nominal sail properties are listed in Table 7.4.

Altogether, a large number of parameters can be varied to define the hull form. Because of the fundamental dimensionality problem of considering too many parameters, only a select few are considered. The typical, nominal geometry uses the parameters listed in Tables 7.2 and 7.3. A diagram of the generic hull form geometric parameters is shown in Figure 7.2.
### 7.2. Approach

#### Table 7.4: Nominal sail properties.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hydrofoil</td>
<td>NACA 0020</td>
</tr>
<tr>
<td>Cap thickness</td>
<td>$t_{cap}/t = 0.25$</td>
</tr>
<tr>
<td>Chord $c/D$</td>
<td>1.0</td>
</tr>
<tr>
<td>Aspect ratio $b/c$</td>
<td>0.5</td>
</tr>
<tr>
<td>LE axial location $x_{LE}/D$</td>
<td>2.0</td>
</tr>
</tbody>
</table>

**Figure 7.2:** Diagram of generic hull form geometric parameters.

#### 7.2.5 INSEAN E1619 Generic Propeller

The INSEAN E1619 propeller functions as a generic marine vehicle propeller. For self-propelled simulations, the propeller diameter $D_p$ is scaled to $D_p/D = 0.5$ and located at $x/L = 0.985$ on the stern of the body. Appearances of the propeller in the open literature are listed in Table 7.1. The propeller has 7 blades, zero rake and moderate skew. Spanwise distributions of chord $c$, thickness $t$, skew angle, and pitch angle are plotted in Figure 7.4. The propeller radius $R_p$ non-dimensionalizes the chord and radial coordinate $r$. These distributions were extracted from a computer-aided design (CAD) geometry, so that they could be referenced by the AL model during CFD simulations.

Advance ratio $J$, thrust coefficient $C_T$, torque coefficient $C_Q$, and efficiency coefficient $\eta$ for
Chapter 7. Parameterized Hull Form: Influence of Cross-Section, Sail Appendage, and Slenderness

Figure 7.3: INSEAN E1619 propeller.

Figure 7.4: INSEAN E1619 spanwise characteristics.
the propeller are defined as,

\[ J = \frac{U_0}{nD_p} \]
\[ C_T = \frac{F_T}{\rho n^2 D_p^4} \]
\[ C_Q = \frac{T_Q}{\rho n^2 D_p^5} \]
\[ \eta = \frac{1}{2\pi} \frac{C_T}{C_Q} J \]

where \( U_0 \) is freestream velocity, \( n \) is rotations–per–second, \( F_T \) is thrust, and \( T_Q \) is torque. These propulsion coefficients are later compared to experimental measurements.

The propeller is simulated in an open–water configuration. Three meshes were studied to ensure grid convergence: a coarse mesh of \( 5.3 \times 10^6 \) cells, a medium mesh of \( 12.6 \times 10^6 \) cells, and a fine mesh of \( 21.3 \times 10^6 \) cells. Cutting planes through the coarse mesh are shown in Figure 7.5.

The magnitude of the instantaneous body force \( ||f_p||/(\rho_0 R_p \text{ rps}^2) \) for the three mesh resolutions is shown in Figure 7.6. The body force is non–dimensionalized by reference density \( \rho_0 \), propeller radius \( R_p \), and rotational speed rps in rotations–per–second. The curved projection of the body force onto the much reflects the skew of the INSEAN E1619 propeller blades. Throughout the unsteady simulation these force distributions rotate. The magni-
tude of the force is based on the sectional airfoil, chord length, local velocity, and local flow angle–of–attack.

Figure 7.6: Non–dimensional body force $||f_p||/(\rho_0 R_p \text{ rps}^2)$ on propeller plane for each mesh resolution.

### 7.2.6 CFD Database

A select few parameters are varied to build the CFD data set. These parameters include hull–form cross–sectional aspect ratio $D_y/D_z$, sail aspect ratio $b/c$, and length–to–diameter ratio $L/D$. Cross–sectional aspect ratio varies from 1.0 to 2.0, sail aspect ratio varies from 0.0 to 0.5, and length–to–diameter ratio is varies from 7.0 to 10.0. Parameter permutations are listed in Table 7.5. In total the data set consists of 40 simulations. For visual clarity, the $b/c = 0.125$ cases are omitted, so only 32 of the 40 cases are shown. These simulations required moderate computational expense. Running on 512 cores, individual simulations could take up to 4 days in real time, depending on the complexity of the particular mesh.

<table>
<thead>
<tr>
<th>Parameter Permutations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cross–sectional aspect ratio $D_y/D_z$</td>
</tr>
<tr>
<td>Sail aspect ratio $b/c$</td>
</tr>
<tr>
<td>Length–to–diameter ratio $L/D$</td>
</tr>
</tbody>
</table>

For $L/D = 10$, the Reynolds number is $Re_L = U_0 L/\nu = 400 \times 10^6$, and for $L/D = 7$, the Reynolds number is $Re_L = U_0 L/\nu = 280 \times 10^6$, where $U_0$ is the ship speed and $\nu$ is the kinematic viscosity. Because the $L/D = 7$ has a smaller hull displacement and surface area, its drag is also lower. Consequently, it will have a different propeller rotation rate to achieve
self-propulsion. The self-propulsion points for both Reynolds numbers were computed iteratively using Newton’s method. Table 7.6 lists several coefficients and ratios computed for the appendageless BOR vehicles for \(L/D = 10\) and \(L/D = 7\). The advance \(J\) reflects the propeller rotation rate computed to achieve self-propulsion. The thrust coefficient \(C_T\), torque coefficient \(C_Q\) and thrust-to-drag ratio \(F_T/F_D\) are also listed. For more complicated geometries, the introduction of the sail appendage and non-BOR hull forms also add additional drag. However, for simplified analysis the self-propulsion points were not computed for each of those cases. Instead the advance ratio of \(J = 1\) was used for all \(L/D = 10\) simulations and \(J = 1.21\) was used for all \(L/D = 7\) simulations.

<table>
<thead>
<tr>
<th>(L/D)</th>
<th>(Re_L)</th>
<th>(J)</th>
<th>(C_T)</th>
<th>(C_Q)</th>
<th>(F_T/F_D)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>(400 \times 10^6)</td>
<td>1.17</td>
<td>0.143</td>
<td>0.0280</td>
<td>1.00</td>
</tr>
<tr>
<td>7</td>
<td>(280 \times 10^6)</td>
<td>1.21</td>
<td>0.107</td>
<td>0.0222</td>
<td>1.00</td>
</tr>
</tbody>
</table>

The simulations consider a linear temperature stratification with constant salinity. The Froude number \(Fr\), which provides a measure of the density stratification, is \(Fr = U_0/ND \approx 205\) with the Brunt–Väisälä frequency, \(N = (1/2\pi) \sqrt{(-g/\rho_0) (\partial \rho / \partial z)} \approx 2 \times 10^{-3}\) s\(^{-1}\). The Richardson number \(Ri\), which is the ratio of buoyancy to flow gradient terms [37], is defined as \(Ri = (g/T_0) (dT/\partial z)/(d\bar{U}/\partial z)^2\). In this equation, \(T_0\) is a reference temperature, \(\bar{T}\) is the mean temperature, and \(\bar{U}\) is the mean velocity. Setting \(dT/\partial z\) to the background stratification intensity \(dT_B/\partial z\), and computing the maximum \(\bar{U}/\partial z\) in the IDP, the Richardson number is found to be \(Ri \approx 2 \times 10^{-2}\). This small magnitude indicates that the propeller-driven near-wake inertial forces dominate the buoyancy forces. Further downstream in the far field, the local velocity will decay, eventually resulting in a large \(Ri\) allowing buoyancy to drive the flow.

Flow fields in the CFD database are non-dimensionalized so that they can be rescaled to other conditions by the designer. Flow axial velocity \(U_x\), horizontal velocity \(U_y\), vertical velocity \(U_z\), temperature perturbation \(\Delta T\), turbulence dissipation rate \(\epsilon\), and turbulence kinetic energy \(k\) are scaled as \((U_x - U_0)/U_0\), \((U_y - U_0)/U_0\), \((U_z - U_0)/U_0\), \(\Delta T/\Delta T_Rp\), \((k - k_B)/U_0^2\), and \((\epsilon - \epsilon_B)/(U_0^3/R_p)\) respectively. The ship speed \(U_0\) and the propulsor radius \(R_p\) scale most fields. The temperature field is non-dimensionalized by the linear change in temperature of the background stratification over the depth of one propulsor radius \(\Delta T_{R_p}\). Background values, \(k_B\) and \(\epsilon_B\), are subtracted from the turbulence quantities. Flow fields are insensitive to Froude number due to large inertial forces. So long as temperature profiles are non-dimensionalized by \(T_{R_p}\), changes in thermal stratification intensity do not change the temperature profile. This means that temperature profiles can be rescaled for different stratification intensities.
7.2.7 Self–Propelled Body Computational Meshes

All computational meshes for the self–propelled cases were built using the Cartesian mesh generation software \textit{cfMesh} [1]. Cells were clustered close to the body, around the sail appendage, around the AL model propulsor region, and in the wake region. Inlet, outlet, and far–field boundaries are located two body lengths away from the body. Simulations with larger domains showed that the boundaries did not affect the wake solution. Certain features were specified for each generated mesh, as shown in Table 7.7. The minimum boundary layer cells were sized to maintain a law–of–the–wall distance of $y^+ < 100$ to capture boundary layer effects and the viscous drag of the hull with the use of wall functions. An example mesh is shown in Figure 7.7 for the axisymmetric case with a sail where $D_y/D_z = 1.0$, $b/c = 0.5$ and $L/D = 10$. Close–up views are shown for the sail, boundary–layer region, and propulsor region where the unsteady propeller AL forces rotate throughout the simulation. Figure 7.8 shows cross–sectional slices through the mesh at various distances downstream. Cells are clustered in the wake of the sail and hull.

<table>
<thead>
<tr>
<th>Mesh feature</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of boundary layer cells</td>
<td>&gt; 20</td>
</tr>
<tr>
<td>Near–wall mesh spacing</td>
<td>$y^+ &lt; 100$</td>
</tr>
<tr>
<td>Wake cells per $D_p$</td>
<td>100</td>
</tr>
<tr>
<td>Wake region extent</td>
<td>$x/L \leq 1.6$</td>
</tr>
<tr>
<td>Total number of cells</td>
<td>$\approx 30 \times 10^6$</td>
</tr>
<tr>
<td>Maximum cell aspect ratio</td>
<td>$AR &lt; 170$</td>
</tr>
<tr>
<td>Maximum cell non–orthogonality</td>
<td>$&lt; 45^\circ$</td>
</tr>
<tr>
<td>Maximum cell skewness</td>
<td>$&lt; 0.8$</td>
</tr>
</tbody>
</table>

7.2.8 Kinetic and Potential Energy

Kinetic and potential energies are computed for each data–set wake profile as,

\[
ke_x = \frac{1}{2} \rho U_x^2, \quad ke_\theta = \frac{1}{2} \rho U_\theta^2, \quad pe = -\frac{1}{2} \frac{g}{\partial \rho_0 / \partial z} (\rho - \rho_0)^2
\]  

\[
KE_x = \iiint_A ke_x \, dA, \quad KE_\theta = \iiint_A ke_\theta \, dA, \quad PE = \iiint_A pe \, dA,
\]  

In the first set of equations, $ke$, and $pe$ are the per–unit–volume energy which are integrated over the plane of the wake to give energy per–unit–length, $KE$ and $PE$. Kinetic energies are computed for individual components of velocity. For example the $KE_x$ only uses the axial velocity component $U_x$, and $KE_\theta$ only uses the azimuthal, “swirl”, component of velocity.
7.2. Approach

Figure 7.7: Side view of the 27.4 million cell mesh extending to $x/L = 1.5$ for the $D_y/D_z = 1$, $b/c = 0.5$, and $L/D = 10$ geometry. Zoomed in views show cells focused in the sail, boundary layer, and propulsor regions.

(a) $x/L = 0.25$. (b) $x/L = 0.50$. (c) $x/L = 0.75$. (d) $x/L = 1.00$. (e) $x/L = 1.25$. (f) $x/L = 1.50$.

Figure 7.8: Cross-sectional view at multiple axial stations of the 27.4 million cell mesh for the $D_y/D_z = 1$, $b/c = 0.5$, and $L/D = 10$ geometry.
Chapter 7. Parameterized Hull Form: Influence of Cross–Section, Sail Appendage, and Slenderness

The potential energy per–unit–volume $pe$ follows the formulation of Holliday and McIntyre [23].

7.3 Results

7.3.1 Open Water Propeller

The AL model is used in the open–water computations of the INSEAN E1619 propeller for advance ratios $J = 0.5$ and $J = 0.65$, and compared to the experiment of Di Felice et al. [14], as shown in Figure 7.9. The computed thrust coefficient shows excellent agreement. The torque coefficient shows good agreement but under–predicts the magnitude found in the experiment. LES predictions from Di Felice et al. [14] similarly under–predicted the magnitude. Because of the under–predicted torque, the efficiency coefficient is slightly over–predicted but otherwise shows good agreement.

![Figure 7.9: INSEAN E1619 open–water propulsion coefficients.](image)

Additional comparison is made to the LDV measurements and LES simulations of Liefvendahl et al. [32] for $J = 0.65$. Measurements and simulations of axial velocity $U_x/U_0$ are shown for $x/R_p = 0.17$, 0.58 and 1.0 in Figures 7.10, 7.11, and 7.12, respectively. These contour plots are clipped at a threshold of one propeller radius. At $x/R_p = 0.17$, The AL model shows differences in its propeller blade wakes when compared to the LDV and LES results of Liefvendahl et al. [32], however the axial momentum distribution along the radial direction is similar. At $x/R_p = 0.58$ and $x/R_p = 1.0$, similar observations are made, however the axial velocity is focused into narrow regions.
7.3. Results

(a) LDV [35].  
(b) LES [35].  
(c) AL model

Figure 7.10: Axial velocity $U_x/U_0$ comparison to LDV and LES of Liefvendahl et al. [35] at $x/R_p = 0.17$, clipped such that $\sqrt{y^2 + z^2} \leq R_p$.

(a) LDV [35].  
(b) LES [35].  
(c) AL model

Figure 7.11: Axial velocity $U_x/U_0$ comparison to LDV and LES of Liefvendahl et al. [35] at $x/R_p = 0.58$, clipped such that $\sqrt{y^2 + z^2} \leq R_p$.

(a) LDV [35].  
(b) LES [35].  
(c) AL model

Figure 7.12: Axial velocity $U_x/U_0$ comparison to LDV and LES of Liefvendahl et al. [35] at $x/R_p = 1.0$, clipped such that $\sqrt{y^2 + z^2} \leq R_p$. 
Chapter 7. Parameterized Hull Form: Influence of Cross–Section, Sail Appendage, and Slenderness

Time–averaged velocity defect profiles for each mesh resolution are shown in Figure 7.13 at three axial locations: $x/R_p = 0.17$, $0.58$, and $1.0$. The unsteady velocity was averaged over the period of one propeller rotation. Near the propeller–hub junction at $r/R_p = 0.2$, some discrepancies are found between mesh resolutions. Otherwise, the velocity defect solution is converged for $r/R_p > 0.2$. This convergence shows that even the coarsest mesh will resolve the AL force distribution.

7.3.2 Case Study: Body of Revolution with Sail Appendage

Given the permutations in $D_y/D_z$, $b/c$, and $L/D$ that compose the 40 cases simulated in this study, it is unrealistic to present extensive details of every one. Instead, special attention is given to the BOR case with a sail where $D_y/D_z = 1.0$, $b/c = 0.5$, and $L/D = 10$. First the flow is visualized around the self–propelled vehicle. A grid convergence study follows to show the appropriateness of the meshing scheme used for all cases. Next, the transition from near to far field wake regions is shown by comparing axial velocity contours downstream of the propulsor. The evolution of the wake is further illustrated through time–averaged vertical profiles of axial velocity and temperature perturbation. Contributions from the upstream sail appendage are discussed.

Flow Visualization

Vortex structures in the flow field are visualized in Figure 7.14 using $Q$–criterion with iso–surfaces taken for $(L/U_0)^2Q = 1.5$ extending to half of a body length downstream where
7.3. Results

Figure 7.14: $Q$–criterion flow visualization of vortex structure with $(L/U_0)^2Q = 1.5$ extending to $x/L = 1.5$ for the BOR geometry with a sail appendage.

$x/L = 1.5$. Counter–rotating vortices are shed from the tip of the sail, extending to the IDP region at $x/L = 1.5$. Potential effects from the hull modify their path. A second pair of vortices is generated directly below the first pair. Near the interface between the sail and the hull a pair of necklace vortices originate and propagate downstream until they are ingested by the propeller and propagate further downstream with alterations due to propulsor swirl. Smaller vortex structures form around the hull as a result of unsteadiness in the simulation and small imperfections in the discrete mesh cells that compose the hull form. At the propeller region, tip vortices originate from the 7 propeller blades. These structures interact with one another and the necklace vortices until they are no longer visible after approximately 3 propeller diameters downstream. Propeller blade root vortices are also generated but are difficult to see in this visualization. A large hub vortex extends from the stern of the hull to the IDP. This flow visualization shows the genesis and evolution of vortex structures due to the vehicle geometry and spinning propeller.

Figure 7.15 shows the instantaneous axial velocity defect $U_x/U_0 - 1$ at the axial locations: $x/L = 0.2, 0.4, 0.6, 0.8, 1.0, 1.3, \text{ and } 1.5$. A velocity deficit is formed due to the sail wake.
The sail tip vortex region is visible as well as the region of necklace vortices formed at the junction of the sail and the hull. The sail wake propagates downstream, growing in size and decaying in magnitude, until the IDP at \( x/L = 1.5 \). At the stern of the body, the propulsor imparts positive momentum onto the surrounding medium. Seven distinct blade wakes are visible at \( x/L = 1.0 \). By \( x/L = 1.5 \), however, the wake is well–mixed and the distinct blade features are no longer visible. Positive velocity from the propeller thrust and negative velocity from hull–form and sail drag are evident at the IDP. The drag wake from the sail appears largely decoupled from the propeller–driven region of the wake.

Figure 7.15: Contours of instantaneous axial velocity defect \( U_x/U_0 - 1 \) at \( x/L = 0.2, 0.4, 0.6, 0.8, 1.0, 1.3, \) and 1.5 for the BOR geometry with a sail appendage.

Grid Convergence Study

To ensure that the mesh could sufficiently resolve the vehicle wake, a grid convergence study was conducted. For this study, Cartesian mesh cells are refined in the wake region behind the sail and stern of the body. Three mesh resolutions are considered as shown in Table 7.8. The coarsest mesh supports 50 cells per vehicle diameter \( D \), the medium mesh supports 100, and the fine mesh supports 200. The medium mesh resolution is used for all other simulations used to build the CFD data set.
7.3. Results

Table 7.8: Grid convergence study mesh refinement Cartesian cell sizes.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>$N_{\text{cells}}/D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>50</td>
</tr>
<tr>
<td>Medium</td>
<td>100</td>
</tr>
<tr>
<td>Fine</td>
<td>200</td>
</tr>
</tbody>
</table>

Figure 7.16 shows the axial velocity defect $U_x/U_0 - 1$ plotted as a function of streamwise position $x/L$ until the IDP location, where $x/L = 1.5$, for the three mesh resolutions. Three vertical positions are considered: along the hull centerline where $z/R_p = 0$, along the sail half-span where $z/R_p = 3$, and along the sail–tip where $z/R_p = 4$. Velocity defect values are negative due to drag from the vehicle. As the wake evolves, the velocity defect approaches the freestream value of zero. The profile along $z/R_p = 0$ follows the wake centerline, within the negative-momentum center region of the NZM wake downstream the propeller hub. The wake here decays at a different rate than behind the sail due to coupling between the positive momentum from the propeller and negative momentum from hull–form drag. The wake behind the sail is largely decoupled from the propeller thereby behaving more as a towed wake. Velocity defect decay behind the sail is affected by the potential effects of the hull displacing the sail wake upward initially and later pulling the sail wake downward due to the adverse pressure gradient at the stern. This phenomenon can be best–visualized by following the path of the sail tip vortices in Figure 7.14, which visualizes vortex structures in the flow. Comparison between the three mesh resolutions demonstrates grid convergence. While the coarse–mesh results show anomalies, the medium and fine meshes converge on a solution.

The vertical distributions of the velocity defect and temperature perturbation at the IDP, where $x/L = 1.5$, are shown in Figure 7.17 for the three mesh resolutions. The negative momentum due to drag from the hull is evident at $z/R_p = 0$. Outside of this region, positive momentum from the propeller is present. Further upward for $z/R_p > 2$, drag effects from the sail are present. Drag regions from the sail tip vortices are located near $z/R_p = 4$. For $z/R_p > 4.5$ the sail wake ends and only the freestream flow is present.

The operating vehicle causes perturbations to the temperature field, which is initially linearly stratified. Potential effects from the hull, swirl from the propeller, and wake entrainment all play a role in redistributing density and temperature fields, resulting in the generation of potential energy due to buoyancy. A large temperature perturbation is seen downstream of the spinning propeller, as shown in Figure 7.17. Positive $\Delta T/\Delta T_{R_p}$ indicates warmer fluid relative to the linear background temperature distribution, while a negative value indicates relatively colder fluid. Swirl from the propeller redistributes $\Delta T/\Delta T_{R_p}$ within approximately one propeller radius of the centerline. Above this region, for $z/R_p > 1.5$, sail wake entrainment also redistributes fluid, generating additional potential energy.
Chapter 7. Parameterized Hull Form: Influence of Cross-Section, Sail Appendage, and Slenderness

Figure 7.16: Axial velocity defect $U_x/U_0 - 1$ for three mesh resolutions of the BOR geometry with a sail appendage at three $z$ locations: tip of the sail, half-span of the sail, and hull centerline.

Figure 7.17: Axial velocity defect and temperature perturbation for three mesh resolutions of the BOR geometry with a sail appendage at $x/L = 1.5$. 
Wake Evolution

The wake evolution of the medium–resolution mesh is studied. Figure 7.18 shows contours of the axial velocity defect $U_x/U_0 - 1$ at the axial planes: $x/L = 1.01, 1.10, 1.20, 1.30, \text{ and } 1.45$. The top row shows the instantaneous field while the bottom row shows the time–averaged field, which is averaged over the time span of one propeller period. Each column shows a separate axial plane. Directly behind the propulsor at $x/L = 1.01$ the seven propeller–blade wake features are visible in the instantaneous case. Discrete propeller–blade features are still visible at $x/L = 1.10$, however for $x/L \geq 1.20$ the features are no longer present due to mixing in the wake. Additionally, the instantaneous and time–averaged fields are equivalent by $x/L = 1.45$, demonstrating near–wake transition to the far wake. At the IDP the wake profile is temporally stationary.

![Figure 7.18: Comparison of instantaneous and time–averaged velocity defect $U_x/U_0 - 1$ at axial stations downstream of the body.](image)

Vertical distributions of the wake profile are shown in Figure 7.19 for the time–averaged velocity defect and temperature perturbations, $U_x/U_0 - 1$ and $\Delta T / \Delta T_{R_p}$. Three downstream locations are considered: $x/L = 1.1, 1.3, \text{ and } 1.5$. The wake velocity defect decays in magnitude most–rapidly in the propeller region. Positive momentum from the propeller thrust and negative momentum from the hull drag are both visible. Velocity in the wake of
the sail decays at a slower rate because it is largely decoupled from the positive momentum of the propeller.

\[ \frac{\Delta T}{\Delta T_{R_p}} \]

Figure 7.19: Vertical profiles of time-averaged velocity defect and temperature perturbation at several axial stations behind the BOR geometry with a sail appendage.

The evolution of \( \Delta T/\Delta T_{R_p} \) is also shown through its vertical distribution in Figure 7.19. In the mixed-patch region behind the propeller, \( \Delta T/\Delta T_{R_p} \) gradually expands in size and decays in magnitude as a consequence of the rapidly changing wake profile. The temperature perturbation within the sail wake instead shows negligible decay between \( x/L = 1.1 \) and \( x/L = 1.5 \). This relative persistence in \( \Delta T/\Delta T_{R_p} \) is a consequence of the sail wake size remaining largely unchanged which displaces the temperature field. Inertial forces from the wake are still relatively large compared to the buoyancy forces.

### 7.3.3 Effect of Cross-Sectional Aspect Ratio and Sail Appendage

Analysis of the 40 simulated cases composed of permutations in \( D_y/D_z \), \( b/c \), and \( L/D \) is presented. The first set of cases presented have \( L/D = 10 \) while the second have \( L/D = 7 \). First, the flow field of axial velocity defect is visualized for a sample of these cases.

Figure 7.20 shows vertical cutting plane view of the instantaneous axial velocity defect \( U_x/U_0 - 1 \) for several hull-form and sail permutations with \( L/D = 10 \). These plots extend to \( x/D = 15 \), which is 5 body diameters or 0.5 body lengths downstream for the \( L/D = 10 \) geometries. The top plot shows a BOR without a sail, where \( D_y/D_z = 1.0 \) and \( b/c = 0.0 \). Potential effects from the hull displacement are visible. Flow accelerates over the front of the body, where there is a positive pressure gradient and decelerates over the stern, where
there is a negative pressure gradient. Flow is then ingested by the propeller and expelled into the wake, providing thrust to the vehicle. An annular region of positive velocity defect is generated, and distinct blade features can be discerned.

The second body of Figure 7.20 introduces a sail with $b/c = 0.25$ onto the BOR hull. The sail is shown to obstruct the flow above the body, leaving behind a deficit of velocity in its wake. The lower portion of the sail wake eventually reaches the stern and interacts with the propeller. The third body shows a larger sail with $b/c = 0.5$. In this case the effects of the sail are more exaggerated. A non-BOR with a sail is considered in the fourth plot with $D_y/D_z = 1.5$. The hull leaves behind a non-circular wake which interacts with the propeller. These effects are exaggerated for the final body with $D_y/D_z = 2.0$. Altogether, these plots help to show the qualitative impact of the geometry features on the near field.

Next, IDP profiles are examined for variations in $D_y/D_z$ and $b/c$ for geometries with slender-ness ratios $L/D = 10$ and $L/D = 7$. Geometries with $L/D = 10$ are examined first. Figure 7.21 shows the maximum cross-sectional extent of the hull-form geometries. Hull forms are equivalent in displacement volume when ignoring the sail appendage. Columns in the figure specify $D_y/D_z$, and rows specify $b/c$. For example, the top left plot shows the axisymmetric BOR. The bottom left plot shows the BOR with a sail of aspect ratio $b/c = 0.5$. The bottom right plot shows the non-BOR $D_y/D_z = 2$ geometry with $b/c = 0.5$. This figure functions as a reference to the underlying geometries used in the following IDP profile figures.

Each subplot title in Figure 7.21 lists the computed drag coefficient $C_D$ for the geometry of both slenderness ratios: $L/D = 10$ and $L/D = 7$. The drag coefficient is computed as $C_D = F_D/(\pi/8 \rho_0 U_0^2 D^2)$, where $F_D$ is the drag force, $\rho_0$ is the background density, and $D$ is the BOR diameter. Predictably, increases in sail appendage span and non-BOR aspect ratios result in a corresponding increase in vehicle drag. Elongation of the hull further increases drag.

Axial velocity defect $U_x/U_0 - 1$ is examined in Figure 7.22 for $L/D = 10$. The top left plot shows the IDP of the axisymmetric BOR. The resulting IDP is also axisymmetric as a result of the geometry. An annular region of positive velocity is generated from propeller thrust, while a negative velocity inner region is generated from hull drag. Looking across the columns, it is evident that non-BOR geometries introduce asymmetry into the IDP. The wake of the hull introduces lateral regions of negative momentum and also interacts with the propeller thrust concentrating regions of positive momentum. Looking down the rows, the effect of the sail appendage is visible. The sail wake adds a vertical region of negative momentum. The sail wake also interacts with the propeller, introducing asymmetry through small regions of concentrated momentum.

At the top of each subplot in Figure 7.22 the integrated kinetic energy due to axial velocity defect is expressed as $10^6 KE_x/(\rho_0 D^2 U_0^2)$. Kinetic energy of the appendageless BOR case decays most rapidly. The non-BOR hulls and the sail modify the wake evolution by adding additional drag, resulting in additional kinetic energy at the IDP. Large regions of negative momentum become decoupled from the propeller thrust.
Figure 7.20: Side view of axial velocity defect $U_x/U_0 - 1$ for several hull-form and sail permutations with $L/D = 10$. 

(a) $D_y/D_z = 1.0$, $b/c = 0.0$
(b) $D_y/D_z = 1.0$, $b/c = 0.25$
(c) $D_y/D_z = 1.0$, $b/c = 0.5$
(d) $D_y/D_z = 1.5$, $b/c = 0.5$
(e) $D_y/D_z = 2.0$, $b/c = 0.5
Figure 7.21: Maximum cross-sectional extent of various hull forms.
Chapter 7. Parameterized Hull Form: Influence of Cross-Section, Sail Appendage, and Slenderness

Figure 7.22: IDP axial velocity defect $U_x/U_0 - 1$ with variations in hull–form and sail aspect ratio for $L/D = 10$. 
The swirl component of velocity $U_{\theta}/U_0$ is considered in Figure 7.23 for $L/D = 10$. For the appendageless BOR case, an axisymmetric swirl profile is found. For the other cases, the effect of the hull form and sail is small, because the generation of swirl is dominated by propeller effects. The hull form and sail still introduce small amounts of asymmetry.

Kinetic energy due to the swirl component of velocity is integrated over the IDP plane and expressed as $10^6 KE_{\theta}/(\rho_0 D^2 U_0^2)$ at the top of each subplot in Figure 7.23. The sail has a negligible effect. Effects from non–BOR hull forms result in a small decrease of $KE_{\theta}$. Interaction between the hull wake and the propeller result in a higher decay of swirl.

Figure 7.24 shows the temperature perturbation $\Delta T/\Delta T_{R_p}$ for each geometry for $L/D = 10$. In all cases background temperature is linearly stratified. The vehicle passage and propeller swirl disturbs this stratification. For the appendageless BOR case the effects of the propeller swirl are clear. Relatively colder fluid is transported from the bottom of the wake upward, while relatively warmer fluid is transported from the top of the wake downward. The effects of the sail and hull–form are small. The sail introduces a small region of relatively colder fluid by entraining fluid from above. Non–BOR hull forms modify the IDP structure near the left and right edges. The $\Delta T/\Delta T_{R_p}$ IDP structure is formed entirely by the flow–field inertial forces. Buoyancy forces are relatively weak in the near field as indicated by the small Richardson number. In the far field, buoyancy forces may become stronger relative to the inertial forces.

Potential energy is integrated over the IDP planes and expressed as $10^6 PE/(\rho_0 D^2 U_0^2)$ at the top of each subplot. Potential energy is smallest for the appendageless cases. The intrusion of the sail into the flow field increases potential energy overall, primarily due to the introduction of tip and necklace vortices which displace the temperature field. As a result, the sail height, itself, does not significantly alter potential energy. The complicated interaction between the root sail wake and the propeller wake appear to non–linearly alter potential energy.

Additional flow fields are presented in Section B of the appendix for axial vorticity $\omega_z R_p/U_0$, turbulent kinetic energy $10^3 (k - k_b)/U_0^2$, and turbulence dissipation rate $10^3 (\epsilon - \epsilon_b)/(U_0^3/R_p)$. Note that although the $k - \omega$ SST turbulence model was employed, $\epsilon$ is presented instead of $\omega$. The turbulence dissipation rate is computed from the simulations as, $\epsilon = C_\mu k^2/\nu_t$, where $C_\mu = 0.09$ and $\nu_t$ is the eddy viscosity. The sail appendage only has a small impact on the axial vorticity. Vorticity is present in the sail tip vortices, but its magnitude is too small to appear within the contour levels of these plots. Non–BOR hull forms result in the generation of vortex structures above and below the wake center. For the turbulence quantities in the IDP, the influence of the hull form and sail are small.

### 7.3.4 Effect of Slenderness

The effects of slenderness in the vehicle geometry are studied by examining the previous geometry permutations but shortening the hull to $L/D = 7$ instead of $L/D = 10$. Figure
Figure 7.23: IDP swirl velocity $U_\theta/U_0$ with variations in hull–form and sail aspect ratio for $L/D = 10$. 
7.3. Results

Figure 7.24: IDP temperature perturbation $\Delta T / \Delta T_{R_p}$ with variations in hull–form and sail aspect ratio for $L/D = 10$. 
Chapter 7. Parameterized Hull Form: Influence of Cross–Section, Sail Appendage, and Slenderness

7.25 shows a vertical cutting plane through the center of several geometries colored by axial velocity defect. These plots extend to $x/D = 12$, which is 5 body diameters downstream. Compared to the previous cases for $L/D = 10$ in Figure 7.20, the flow field around the body is shortened. Interactions between the hull wake and the propeller appear more pronounced. Otherwise, the flow is qualitatively similar.

The $L/D = 7$ IDP axial velocity defect is shown in Figure 7.26. Flow structures are similar to that of the $L/D = 10$ case, however some differences are notable. For $L/D = 7$, the total drag from the hull is smaller requiring less propeller thrust for self–propulsion. As a result, the velocity defect magnitude is also smaller. Non–BOR hulls for $L/D = 7$ also leave a smaller region of negative momentum. Because of these two differences, integrated kinetic energy is lower for the $L/D = 7$ case.

Swirl velocity for $L/D = 7$ is shown in Figure 7.27. In comparison to the $L/D = 10$ cases, these profiles are qualitatively similar but have more–distinct features. Kinetic energy due to swirl is smaller because the propeller operates at a lower power setting.

The temperature perturbation field for the $L/D = 7$ cases is shown in Figure 7.28. Overall, the mixed–patch region is smaller in size resulting in less potential energy as compared to the $L/D = 10$ cases. However, the increase in potential energy due to non–BOR hull–form effects is more distinct for the $L/D = 7$ cases.

Additional flow fields for $\omega_x R_p/U_0$, $10^3(k - k_b)/U_0^2$, and $10^3(\epsilon - \epsilon_b)/(U_0^3/R_p)$ for $L/D = 7$ are presented in Section B of the appendix. Axial vorticity for the $L/D = 7$ cases is more–affected by geometry parameters than for $L/D = 10$. The tip vortices are also visible within the color bar levels. Turbulence quantities for $L/D = 7$ are smaller in magnitude. Otherwise, qualitative features are similar between the two sets of cases.

7.4 Conclusions

The linearly–stratified near field was investigated for a survey of self–propelled marine vehicle geometries. Hull–form cross–sectional aspect ratio, sail aspect ratio, and slenderness were all varied to compose the 40 unsteady RANS simulations run at Reynolds numbers of $Re_L = 400 \times 10^6$ and $Re_L = 280 \times 10^6$ based on the hull length. The INSEAN E1619 generic propeller was implemented using the AL method, with open–water simulations showing agreement to the overall momentum distributions of the experimental data. Analysis of the near–field results led to several conclusions. The non–circular drag wakes cast off of non–BOR hull forms interact with the propeller flow generating concentrated regions of positive and negative momentum, increasing kinetic energy. Drag from the sail appendage results in a region of negative momentum that decays largely independently of the propeller wake. Necklace vortices and obstructed flow near the base of the sail, however, do interact with the downstream propeller, modifying the IDP profile. Tip vortices from the propeller redistribute a small amount of fluid, which contributes to the IDP potential energy. The
7.4. Conclusions

Figure 7.25: Side view of axial velocity defect $U_x/U_0 - 1$ for several hull-form and sail permutations and $L/D = 7$. 
Figure 7.26: IDP axial velocity defect $U_x/U_0 - 1$ with variations in hull–form and sail aspect ratio for $L/D = 7$. 
7.4. Conclusions

Figure 7.27: IDP swirl velocity $U_\theta/U_0$ with variations in hull–form and sail aspect ratio for $L/D = 7$. 
Chapter 7. Parameterized Hull Form: Influence of Cross–Section, Sail Appendage, and Slenderness

Figure 7.28: IDP temperature perturbation $\Delta T/\Delta T_R$ with variations in hull–form and sail aspect ratio for $L/D = 7$. 
7.4. Conclusions

effect of hull slenderness is small. Geometries with \( L/D = 10 \) and \( L/D = 7 \) both share the same qualitative flow features, however differences in size and self-propulsion point lead to quantitative differences, such as lower potential and kinetic energies for the \( L/D = 7 \) case.
Bibliography


Appendix B  Contour Maps of Additional Flow–Field Variables

Contour maps are presented for the additional flow–field variables: axial vorticity $\omega_x R_p/U_0$, turbulent kinetic energy $10^3(k-k_b)/U_0^2$, and turbulence dissipation rate $10^3(\epsilon-\epsilon_b)/(U_0^3/R_p)$. All quantities are non–dimensionalized by the propeller radius and freestream velocity. The turbulence quantities have background values subtracted from them, where $k_b$ and $\epsilon_b$ are the background turbulent kinetic energy and turbulence dissipation rate, respectively. Figures B.1, B.2, and B.3 show the flow fields for the $L/D = 10$ geometries, while Figures B.4, B.5, and B.6 show the flow fields for the $L/D = 7$ geometries. The maximum cross–sectional extent of the vehicle for each contour map can be referenced in Figure 7.21.
Figure B.1: IDP axial vorticity $\omega_{x}R_{p}/U_{0}$ with variations in hull-form and sail aspect ratio for $L/D = 10$. 
Figure B.2: IDP turbulent kinetic energy $10^3(k - k_b)/U_0^2$ with variations in hull–form and sail aspect ratio for $L/D = 10$. 
Figure B.3: IDP turbulence dissipation rate $10^3(\epsilon - \epsilon_b)/(U_0^3/R_p)$ with variations in hull–form and sail aspect ratio for $L/D = 10$. 
B. Contour Maps of Additional Flow–Field Variables

Figure B.4: IDP axial vorticity \( \omega_x R_p/U_0 \) with variations in hull–form and sail aspect ratio for \( L/D = 7 \).
Figure B.5: IDP turbulent kinetic energy $10^3(k - k_b)/U_0^2$ with variations in hull–form and sail aspect ratio for $L/D = 7$. 
B. Contour Maps of Additional Flow–Field Variables

Figure B.6: IDP turbulence dissipation rate $10^3(\epsilon - \epsilon_b)/(U_0^3/R_p)$ with variations in hull–form and sail aspect ratio for $L/D = 7$. 
Chapter 8

Deep Learning of the Near Wake with Implicitly–Learned Hull–Form Parameters

Abstract

The high–fidelity analysis of a self–propelled marine vehicle involves evaluating its wake profile from a computationally–expensive, near–field Computational Fluid Dynamics (CFD) simulation. Data–driven deep learning is applied to expedite this task with the use of convolutional neural networks (CNNs) in two approaches. In the first approach, marine vehicle geometric parameters are explicitly specified as the input to the CNN, which then generates the corresponding wake profile. In the second approach, the geometry itself is input as a tensor of discrete volume fractions allowing the network to implicitly learn the underlying geometric parameters. The second, implicit–parameter approach uses a conditional Generative Adversarial Network (cGAN) with a CNN U–Net generator and a CNN discriminator. The end goal of this approach is to evaluate vehicle geometries, even outside of the scope of the training data and explicit parameter space. Both approaches show good performance in inference and prediction, although some anomalous distortions are present in the generated wake profiles. The implicit–parameter approach shows promise for generating wake profiles for arbitrary, non–parameterized geometries, however additional training data is required to achieve acceptable performance for such cases. Even so, both approaches can generate realistic wake profiles within the explicit parameter space with evaluations taking a fraction of a second, successfully allowing for rapid use.
8.1 Introduction

The contemporary analysis of a self–propelled marine vehicle considers the unsteady near–field flow through the use of Computational Fluid Dynamics (CFD). More specifically, a cross–plane wake profile downstream of the vehicle is evaluated. High fidelity near–field simulations are expensive, requiring large time commitments and computational resources to realize a solution. These simulations are prohibitively expensive for rapid use, such as during the conceptual design phase, in which numerous concepts are considered. Sometimes thousands of concepts are evaluated within the paradigms of set–based–design or multi–disciplinary design optimization. Machine learning provides an avenue to accelerate the analysis process by learning the functional relationship between geometric parameters and their corresponding wake profile. Sometimes referred to as “universal approximators”, machine learning neural networks have the capacity to learn the nonlinear relationships of virtually any problem provided they have enough depth.

In this study, two deep learning convolutional neural networks (CNNs) are formulated and trained to learn the functional mapping between vehicle geometry and corresponding wake profile. Wake profiles for this study are taken from the Reynolds–Averaged Navier Stokes (RANS) simulations of a generic marine vehicle geometry presented in Chapter 7 with 40 training cases and 3 additional validation cases. A typical marine vehicle geometry is shown in Figure 8.1. The first network uses fractionally–strided convolutions to generate the wake profile from explicitly–specified geometric parameters. The second network follows a more–advanced approach using a conditional Generative Adversarial Network (cGAN), where the generator is a modified version of the U–Net architecture, originally implemented by Ronneberger et al. [46], to implicitly learn the underlying geometric parameters of the marine vehicles in the training database. The network takes as input a tensor of geometric volume fractions that describe the spatial form of the vehicle. The goal of this network is to develop the capacity to learn numerous geometric parameters from the training set, allowing for the rapid evaluation of arbitrary, non–conventional geometries. Both the explicit– and implicit–parameter networks perform well in inference and prediction. The implicit–parameter approach is further evaluated for a non–conventional vehicle geometry that cannot be described by the explicit geometric parameters of the training data set geometries. Finally, conclusions are made.
204

Chapter 8. Deep Learning of the Near Wake with Implicitly–Learned Hull–Form Parameters

8.2 Related Work

Steady advances in high–performance computing have allowed for larger, more effective machine learning networks. With larger numbers of hidden layers, these “deep”–learning networks have seen widespread success in a broad range of applications such as image recognition [19], medical image analysis [51] and even CFD [30]. Several authors have recently reviewed deep learning [9, 15, 32, 48], however the field continues to advance at a rapid pace.

Convolutional neural networks (CNNs) are well–suited for tasks involving images such as machine vision, because they have the capacity to learn spatial features [16]. LeCun et al. [31] first applied a deep CNN to the image recognition of hand–written numbers in 1998. Since then, the steady improvement in high–performance computing and the widespread availability of training databases such as ImageNet [7] have allowed researchers to effectively apply CNNs to modern computer vision problems. In the 2010 ImageNet competition, Krizhevsky et al. [29] successfully trained a CNN to the classification of 1.2 million high resolution images, demonstrating significant improvement in performance over the previous state–of–the–art and inspiring the widespread use of CNNs in the field. Winning the 2014 ImageNet competition, Simonyan and Zisserman [52] trained a CNN with 16–19 layers, showing that deeper networks with small $3 \times 3$ convolutional kernals improved upon shallower networks with larger kernal sizes. Iandola et al. [22] built the SqueezeNet CNN, which demonstrated similar accuracy to AlexNet but with greater efficiency, using 50 times fewer parameters.

Long et al. [35] successfully applied a fully convolutional network to the problem of image segmentation, in which images are divided into labeled regions of the objects that compose
8.2. Related Work

Badrinarayanan et al. [3] showed how an encoder–decoder CNN framework could encode low resolution feature maps of the image, which could then be upsampled through the decoder into a high resolution image. Chen et al. [4] introduced recent advancements including improvements to upsampling. Because of their visual portrayal, some encoder–decoder CNNs are referred to as “U–Nets”. Skip connections between the encoder and decoder paths facilitate information propagation in these networks. U–Nets have been successful in a range of applications including biomedical imaging.

Coining the term in 2015, Ronneberger et al. [46] first applied a U–Net to 2D biomedical image segmentation. The network structure allowed the contracting path of the U–Net to capture context within the images, while the expanding path localized the discerned features. Çiçek et al. [6] expanded this U–Net to work with 3D images. Li et al. [33] recently proposed a hybrid 2D–3D network to efficiently reduce computational cost. Milletari et al. [40] formulated a similar network to process volumetric data, which they called a V–Net. Jin et al. [26] applied a U–Net to the reconstruction of medical images. U–Nets have been applied by many other researchers to biomedical imaging [10, 21, 44, 47, 49, 71]. They have also been applied to a range of other applications.

In the application of image–to–image translation, Isola et al. [25] applied the U–Net architecture to the generator portion of their cGAN [14, 41]. These adversarial networks work by training a competing generator and discriminator that seek to outperform one another. U–Nets have been used to identify roads and other objects from aerial images [23, 24, 70]. Shah et al. [50] showed how stacking U–Nets within a larger network can aid in image segmentation. Tang et al. [56] trained a U–Net to localize human poses and face key points in images. Han and Ye [17] sought to improve upon U–Net limitations. In a similar approach to the present work, Thuerey et al. [57] applied a modified U–Net architecture to the inference of the steady flow field around an airfoil using RANS CFD data. By coupling the physical domain to the output network solution, they showed how the U–Net architecture is well–suited for simulation–based problems that use Cartesian grids. Using an encoder–decoder framework, Nie et al. [43] also input a Cartesian grid into their network to predict the stress field over a material in deformation. Other researchers have applied deep learning to physics–based fluid flow problems.

Using an encoder–decoder framework, Farimani et al. [12] trained cGANs over solutions of the steady–state heat equations and also the incompressible Navier–Stokes fluid flow equations. With the adversarial cGAN approach, they showed that the network was able to predict solutions outside of the scope of their training data. Zhang et al. [69] used a CNN to infer the lift coefficients over airfoils that were projected onto Cartesian grids as input. Singh et al. [53] studied flow separation over airfoils. Deng et al. [8] optimized the flow over an airfoil using an encoder–decoder framework. Using data–driven approaches, researchers have tried to model the fluid–flow equations directly. Yang et al. [67] and Tompson et al. [58] trained CNNs to approximate the incompressible Euler fluid–flow equations based on numerically computed solutions. Wiewel et al. [63] examined unsteady flows. Chu and Thuerey [5] used CNNs to learn descriptive features to generate smoke flows. Xie et al. [66] applied a GAN
to realistically synthesize unsteady flow. Um et al. [60] applied a neural network to the imitation of splashing flows. Ströfer et al. [55] used a CNN to classify regions of a fluid flow and extract desired flow features. Given the high uncertainties present in solving the RANS equations [65], data–driven approaches have also been applied to improving fluid–flow turbulence modeling [11, 27, 34, 38, 61, 62, 64]. Maulik et al. [39] and Yang et al. [68] trained neural networks to account for sub–grid scale effects in large–eddy simulations. Long et al. [36] developed a network to learn the underlying model of partial differential equations, recently applying their network to Burger’s equation [37].

8.3 Approach

Two CNN networks are discussed and named as “explicit–parameter” and “implicit–parameter” networks because of how they handle input parameters. At a high level, the explicit–parameter CNN in this study can be considered a black–box function with explicitly–specified hull–form geometric parameters \( Z_p = [z_1, z_2, z_3, ..., z_p] \) that are input and the corresponding wake profile is output.

\[
\text{wake profile} = f(Z_p) \quad (8.1)
\]

So, the wake profile can be generated and evaluated rapidly for an arbitrary \( Z_p \). For this study, three geometric parameters of a self–propelled marine vehicle are considered: the hull–form cross–sectional aspect ratio \( D_y/D_z \), the sail aspect ratio \( b/c \), and the length–to–diameter ratio \( L/D \). Therefore, \( Z_p = [D_y/D_z, b/c, L/D] \).

The implicit–parameter CNN, however, does not take precise geometric parameters as input. Instead, the 3D vehicle geometry, itself, is input, expressed as a tensor of volume fractions within a 3D Cartesian domain. The network must then implicitly identify relevant geometric parameters from the training data so that when a new, non–training geometry is input, the proper wake profile can be generated.

\[
\text{wake profile} = f(\text{geometry}) \quad (8.2)
\]

Both networks were built in and trained using the PyTorch framework [1]. Details for them are presented.

8.3.1 Explicit–Parameter Network Structure

The explicit–parameter CNN used in this study is visualized in Figure 8.2. A sequence of fractionally–strided convolutions spatially reshape the input geometric parameter array \( Z_p \) into a \( n_{\text{fields}} \times 80 \times 112 \) cell data plane that contains the wake profiles. Here, \( n_{\text{fields}} \) is the number of flow fields, which is the three components of the velocity defect: \( U_x/U_0 - 1, U_y/U_0, \) and \( U_z/U_0 \) non–dimensionalized by the freestream velocity \( U_0 \). The stream–wise
8.3. Approach

velocity is $U_x$ while the horizontal and vertical transverse components are $U_y$ and $U_z$. Each block within the figure represents a layer in the network and is described by spatial width and height in the lateral directions, and the number of channels in the depth–wise direction. For example, the first block has $5 \times 7$ transverse spatial dimensions and 256 channels. Layers with narrower spatial dimensions have a larger number of channels to facilitate learning. Convolutional weights are learned iteratively through the ADAM optimization scheme [28], and they are initialized through Xavier initialization [13]. The CNN has a total of 717,031 learnable parameters and a state size of 161,283.

![Figure 8.2: Diagram of the CNN with explicitly–specified geometric parameters.](image)

From the input layer to the output layer, the CNN takes the following fractionally–strided convolution operations, where “Conv” corresponds to the convolution operation.

- Conv 1: fractionally–strided, $5 \times 7$ kernal, $0 \times 0$ zero–padding, $1 \times 1$ strides, PReLU activation. Instance normalization is applied between the first and second layers to improve training [59].

- Conv 2-4: fractionally–strided, $4 \times 4$ kernal, $1 \times 1$ zero–padding, $2 \times 2$ strides, PReLU activation.

- Conv 5: fractionally–strided, $4 \times 4$ kernal, $1 \times 1$ zero–padding, $2 \times 2$ strides, tanh activation.

Dropout regularization [54] was tested but did not improve results so is not included in the final network. Training data for each flow field is pre–processed by normalizing by the maximum magnitude. This ensures that all flow–field data varies between -1 and 1, within the bounds of the final tanh activation function. The reverse normalization operation is applied to the output flow field. The Parametric Rectified Linear Unit (PReLU) activation function [18] is used for all activations except for the final layer, which uses the hyperbolic
tangent function \( \tanh \). For an input \( x_I \), the PReLU activation function is,
\[
PReLU(x_I) = \begin{cases} 
  x_I & x_I \geq 0 \\
  ax & x_I < 0 
\end{cases}
\]
with a learnable slope parameter \( a \). Nonlinearity is present in the function due to the discontinuity at 0. The PReLU function was chosen over the conventional Rectified Linear Unit (ReLU) \([42]\), because it effectively avoided dead activations that otherwise appeared when using ReLUs. The sloped curve described by \( a \) allows the network to back-propagate gradients even when negative inputs are given to the PReLU.

Hyperparameters are listed in Table 8.1. The learning rate is initially set to \( \eta_r = 10^{-4} \) for the first 3,000 epochs. It then is set to \( \eta_r = 10^{-5} \) until epoch 30,000, and finally \( \eta_r = 10^{-6} \) for epochs beyond 30,000. Incrementally reducing the learning rate reduced numerical oscillations and numerical instabilities in the learning process. The \( \beta_1 \) and \( \beta_2 \) of the ADAM optimization scheme are set to 0.5 and 0.999 respectively.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Learning rate ( \eta_r )</td>
<td>( 10^{-4}, 10^{-5}, 10^{-6} )</td>
</tr>
<tr>
<td>ADAM ( \beta_1, \beta_2 )</td>
<td>0.5, 0.999</td>
</tr>
</tbody>
</table>

### 8.3.2 Implicit–parameter Network Structure

**Implicit Geometric Parameters**

The implicit–parameter CNN takes the geometry of the marine vehicle, itself, as input so as to implicitly learn the vehicle geometric parameters. The geometry is broken into a tensor of volume fractions over a 3D Cartesian domain. This domain is shown in Figure 8.3 with an example vehicle described by \( D_y/D_z = 1.0, b/c = 0.5, \) and \( L/D = 10 \). In this example, the hull is a body–of–revolution (BOR) as implied by \( D_y/D_z = 1.0 \). The domain extends 12.5 diameters \( D \) in the stream–wise \( x \) direction, 1.6\( D \) in the transverse \( y \) direction, and 2.24\( D \) in the vertical \( z \) direction. The grid has \( 5 \times 40 \times 56 \) cells, where the 5 cells were chosen in the stream–wise direction to adequately resolve changes in the position and size of the sail appendage and changes in overall hull shape. The \( 40 \times 56 \) transverse cells were selected to spatially match the resolution of the wake profile one layer from the final output in the U–Net.

For every cell that the vehicle geometry intersects, the volume fraction of the vehicle volume within that cell is computed. An example of the extracted volume fractions for the mentioned
8.3. Approach

geometry is shown in Figure 8.4. Volume fractions within each of the 5 stream–wise planes reflect the shape of the vehicle. Near the bow and stern, fractions are less than 1 due to hull curvature. Most volume fractions within the parallel midbody encompass the entire cell and are therefore computed as 1. The sail appendage intersects the first two sets of cells in the domain. Due to the small size of the sail, the corresponding volume fractions are small.

A second example is shown in Figure 8.5 for the geometry described by \( \frac{D_y}{D_z} = 2.25, \ b/c = 0.55, \) and \( L/D = 11 \). In this case the non–BOR exhibits non–circular volume fraction distributions to reflect the hull cross–section. Because the hull is longer than in the previous example, the magnitude of the stern volume fractions increases. The sail is also taller than in the previous geometry.

Volume fractions were computed programmatically with a custom Python script interfacing with the FreeCAD software [2]. For each cell within the domain, a 3D block of the cell size was intersected with the solid vehicle geometry at the cell location. Using the custom script, volume fraction tensors can be computed for arbitrary computer–aided design (CAD) files.
Chapter 8. Deep Learning of the Near Wake with Implicitly–Learned Hull–Form Parameters

Figure 8.5: Example of computation of geometric volume fractions for the $D_y/D_z = 2.25$, $b/c = 0.55$, and $L/D = 11$ geometry.

Conditional Generative Adversarial Network

The cGAN used in this study is shown in Figure 8.6. Geometries are input implicitly as tensors of volume fractions. The generator must implicitly learn the underlying geometric parameters. Within the adversarial network, the generator and the discriminator compete with one another. The discriminator learns to discern between real and fake wake profiles by assessing those from the training data set and those from the generator. The generator seeks to fool the discriminator by generating “fake” wake profiles indistinguishable from the “real” training data. Note that realism is entirely dependent on the training data set, which in this case is extracted from RANS CFD simulations. So, the quality of the training data is a fundamental source of uncertainty.

In order to generate the fake wake profiles, non–training, fake geometries must be input into the generator. These fake geometries allow the discriminator to evaluate cases other than those in the training data. The naive approach for formulating these fake geometries would be to input the $5 \times 40 \times 56$ volume fraction tensor with randomized values. However, given the large degrees of freedom within the tensor, the likelihood of randomly finding meaningful geometries is negligible. Instead, the explicit parameters $D_y/D_z$, $b/c$, and $L/D$ are randomized between the ranges $[1.0, 2.25]$, $[0.0, 0.55]$, and $[7.0, 11.0]$ to formulate random geometries, which are converted to input volume fraction tensors. Additional unconventional geometries are also provided to condition the cGAN to cases outside of the scope of the training data.
8.3. Approach

Figure 8.6: Overall cGAN layout: the discriminator learns from training and generated wake profiles, while the generator tries to fool the discriminator. Geometries are input as volume fraction tensors, so the underlying geometric parameters must be learned implicitly.

Adversarial Method

The objective function \( L \) of network includes the cGAN adversarial loss \( L_{cGAN}(G, D) \) [14, 41],

\[
L_{cGAN}(G, D) = \mathbb{E}_{X,Y} \left[ \log D(X, Y) \right] + \mathbb{E}_{X} \left[ \log(1 - D(X, G(X))) \right]
\]  
(8.4)

where \( G \) is the generator function, \( D \) is the discriminator function, \( X \) is the input, and \( Y \) is the output. For this study \( X \) represents the input geometry and \( Y \) the output wake profile. The generator competes with the discriminator by trying to minimize the adversarial loss, while the discriminator tries to maximize it. So, the initial objective \( L \) is the following.

\[
L = \arg \min_{G} \max_{D} L_{cGAN}(G, D)
\]  
(8.5)

An additional reconstruction loss term is included in the objective function to help the optimization to drive towards solutions close to the training data. Pathak et al. [45] has demonstrated benefits of using a reconstruction loss in addition to the adversarial loss. For the present work the reconstruction loss \( L_{L2} \) is set to the \( L_{2} \) norm between generated wake profiles from \( G(X) \) and the ground truth training profiles.

\[
L_{L2} = \mathbb{E}_{X,Y} ||G(X) - Y||^2
\]  
(8.6)

The addition of this term helps to drive the generator close to the ground truth data to keep training error small. The complete objective function is therefore,

\[
L = \arg \min_{G} \max_{D} L_{cGAN} + \lambda_{L2} L_{L2}
\]  
(8.7)
where $\lambda_{L2}$ controls the relative importance of the reconstruction loss to the adversarial loss.

**U–Net Generator within Adversarial Network**

The implicit–parameter generator network is visualized in Figure 8.7. This type of network is referred to as a U–Net because of the visual similarities to its namesake. In this case, a series of convolutions spatially reshape the input volume fractions until they are a 1D array with a batch length of 64. This structure provides the network with the capability of encoding latent geometric parameters. The latent parameters are then reshaped and decoded through fractionally–strided convolutions until they match the wake dimensions of $n_{\text{fields}} \times 80 \times 112$, where $n_{\text{fields}} = 3$ for the three components of the velocity defect. Importantly, the U–Net takes the extra step of connecting the encoding and decoding portions of the network. Activation tensors from the encoding side are concatenated channel–wise onto the decoding activations. Such connections are advantageous for two primary reasons. First, the distribution of volume fractions has physical significance on the wake profile, since the wake shape is correlated to the cross–sectional geometry shape. Second, the connections provide a means for error to back–propagate quickly through the network. Overall, the network is improved in its effectiveness, efficiency, and ease of training for this type of problem. This U–Net has a 176,367 learnable parameters and a state size of 81,827. Other properties are similar to the explicit–parameter network.

Similar to the explicit–parameter network, this network uses PReLU functions after every layer except the final layer, which uses the hyperbolic tangent function $\tanh$. Instance normalization is applied between the first and second layers. This normalization implementation was found to reduce training time and improve predictive performance by reducing over–fitting in the results. The ADAM optimization scheme is employed during the optimization process, and weights are initialized through Xavier initialization. Training data for each flow field was also pre–processed in the same way as the explicit–parameter network by normalizing by the maximum magnitude. Output wake profiles follow the reverse normalization operation. Hyperparameters are listed in Table 8.2. The learning rate is initially set to $\eta_r = 10^{-4}$ for the first 10,000 epochs. It then is set to $\eta_r = 10^{-5}$ until epoch 100,000, and finally $\eta_r = 10^{-6}$ for epochs beyond 100,000. Incrementally reducing the learning rate reduced numerical oscillations and numerical instabilities in the learning process. Like in the previous network, $\beta_1$ and $\beta_2$ of the ADAM optimization scheme are set to 0.5 and 0.999 respectively.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Learning rate $\eta_r$</td>
<td>$10^{-4}, 10^{-5}, 10^{-6}$</td>
</tr>
<tr>
<td>ADAM $\beta_1, \beta_2$</td>
<td>0.5, 0.999</td>
</tr>
</tbody>
</table>
Figure 8.7: Diagram of the U–Net generator for the cGAN with implicitly–learned geometric parameters.
Discriminator within Adversarial Network

The discriminator is a fully convolutional network that estimates the probability that a wake profile is real or not. The structure of this CNN is visualized in Figure 8.8. The wake profile of dimensions $n_{fields} \times 80 \times 112$ is input first. Volume fractions of the corresponding vehicle geometry are input into the second layer through channel-wise concatenation. These volume fractions condition the discriminator’s evaluation for a particular geometry. A series of convolutions reshape the layers until it is output as a single value ranging between 0 and 1. An output of 0.7, for example, corresponds to 70% certainty that the wake profile is real. The discriminator network has a total of 704,229 learnable model parameters and a state size of 172,480.

![Diagram of the discriminator CNN for the cGAN.](image)

**Figure 8.8:** Diagram of the discriminator CNN for the cGAN.

### 8.3.3 CFD Database

The CFD database contains wake profiles extracted from the near-field CFD simulation of a self-propelled generic marine vehicle as described in Chapter 7. Cross-sectional aspect ratio, sail aspect ratio, and length-to-diameter parameters were all varied to compose the 40 training cases of the data set. The number of training cases is small for a typical machine learning problem because of the computational expense required to build each case. Each simulation could take up to 4 days of real time running in parallel on 512 cores. The permutations in geometric parameters to compose the training data set are listed in Table 8.3.

Three additional simulations, not included in the training data set, were also conducted to
Table 8.3: Parameter permutations composing the 40 cases in the CFD data set.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Permutations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cross-sectional aspect ratio (D_y/D_z)</td>
<td>1.0, 1.25, 1.5, 2.0</td>
</tr>
<tr>
<td>Sail aspect ratio (b/c)</td>
<td>0.0, 0.125, 0.25, 0.375, 0.5</td>
</tr>
<tr>
<td>Length-to-diameter ratio (L/D)</td>
<td>7.0, 10.0</td>
</tr>
</tbody>
</table>

evaluate network performance. To study inference, the wake profile was computed for a vehicle with \(D_y/D_z = 1.75\), \(b/c = 0.45\), and \(L/D = 8.5\). To study prediction, the wake profile was computed for a vehicle with \(D_y/D_z = 2.25\), \(b/c = 0.55\), and \(L/D = 11.0\). Lastly, the wake profile of an unconventional geometry was computed to evaluate the implicit-parameter network’s performance for an arbitrary vehicle that could not be described by the explicit geometric parameters. This vehicle has a standard sail size of \(b/c = 0.5\) and standard length of \(L/D = 10\), however the cross-section is asymmetric, with \(D_y/D_z = 1.0\) on the port side and \(D_y/D_z = 2.0\) on the starboard size. The vehicle geometry and associated volume fractions are shown in Figure 8.9.

Figure 8.9: Computation of geometric volume fractions for an unconventional geometry with \(b/c = 0.5\), and \(L/D = 10\), \(D_y/D_z = 1.0\) on port side, and \(D_y/D_z = 2.0\) on starboard side.

8.4 Results

Both the explicit- and implicit-parameter networks are analyzed in performance. Training performance is examined first. Next, the networks are compared in inference and prediction for geometries not included in the training data set. Finally, the performance of the implicit-parameter network is considered for an unconventional geometry that cannot be described
by the explicit geometric parameters and is outside of the scope of the training data. Error is evaluated by direct comparison between generated and ground-truth flow fields and also by comparing integrated kinetic energy.

These three performance evaluation cases are visualized in Figure 8.10. The inference case considers a geometry within the bounds of the training data set, where the geometry is described by $D_y/D_z = 1.75$, $b/c = 0.45$, and $L/D = 8.5$. The prediction case considers a geometry outside of the bounds of the training data set, where the geometry is described by $D_y/D_z = 2.25$, $b/c = 0.55$, and $L/D = 11.0$. The unconventional geometry case cannot be expressed to a network by the three explicit parameters alone and is beyond the scope of the training data. This geometry has an asymmetric cross-section with $D_y/D_z = 1.0$ on the port side and $D_y/D_z = 2.0$ on the starboard side. The sail and slenderness are defined by $b/c = 0.5$, and $L/D = 10$. This figure also lists the drag coefficients for the self-propelled vehicles, $C_D = F_D/(\pi/8 \rho_0 U_0^2 D^2)$, computed from the associated CFD simulations, where $F_D$ is the drag force, $\rho_0$ is the reference density, $U_0$ is the freestream speed, and $D$ is the vehicle diameter.

![Figure 8.10: Three evaluation cases: inference, prediction, and an unconventional geometry outside of the scope of the training data.](image)

### Inference
- $D_y/D_z = 1.75$
- $b/c = 0.45$
- $L/D = 8.5$

### Prediction
- $D_y/D_z = 2.25$
- $b/c = 0.55$
- $L/D = 11$

### Unconventional
- $D_y/D_z = 1.0$ (port)
- $D_y/D_z = 2.0$ (starboard)
- $b/c = 0.5$
- $L/D = 10$

$C_D = 0.0652$

$C_D = 0.0879$

$C_D = 0.0641$

8.4.1 Performance in Training

The iterative training of the explicit-parameter CNN is shown in Figure 8.11. Training error is computed as the mean-squared-error (MSE) of generated wake profiles compared to the training data set. Test error is the MSE of generated wake profiles when compared to a test case not included in the training data. Both errors are non-dimensionalized by the initial training MSE. Both training and test error are shown to decrease dramatically then level off. The test error levels off relatively quickly, which shows that network performance in terms of MSE is quickly established. Additionally, the test error does not increase, which suggests that the network is not over-fitting the training data. The total training time was
approximately 16 days using 1 Intel Skylake Xeon Gold 3 Ghz CPU. Graphical Processing
Unit (GPU) acceleration was not employed in this training.

Figure 8.11: Explicit–parameter CNN training and test error.

The iterative pre–training of the U–Net is shown in Figure 8.12. Both training and test
error are shown to decrease dramatically initially. The test error increases early on and then
gradually decreases, suggesting that over–fitting is not an issue. The total training time was
approximately 3 days using 1 Intel Skylake Xeon Gold 3 Ghz CPU.

Figure 8.12: Pre–training of U–Net generator showing training and test error.

Using the pre–trained U–Net, the overall cGAN is trained. Figure 8.13 shows the iterative
loss of both the generator and discriminator as well as the test error. The $\lambda_{L2}$ parameter
Chapter 8. Deep Learning of the Near Wake with Implicitly-Learned Hull-Form Parameters

is set to $\lambda_{L_2} = 0.1/||G(X)||_2$ using the $L_2$ error of the pre-trained U-Net to ensure that the network continues to adequately capture the training data. The generator attempts to fool the discriminator by generating more-realistic wake profiles. The discriminator seeks to discern between the fake wake profiles from the generator and the true wake profiles from the training set. By competing with one another, manifested in their rapid, opposing oscillations, they both improve iteratively. The figure also shows the test error during adversarial training. The test error decreases as the generator improves in quality and eventually levels off when the network can no longer improve in quality. The discriminator loss gradually decreases, while the generator loss increases, showing the generator becoming less capable of fooling the discriminator. With the generator incapable of improving further, training is halted.

Figure 8.13: Adversarial training losses of the implicit-parameter cGAN generator and discriminator compared to test error.

Generated profiles for 32 of the 40 trained networks are shown in the appendix. Evaluation of either trained network for a single case takes a fraction of a second. Wake profile contours of axial velocity defect generated from the explicit-parameter network are shown in Figure C.1 for $L/D = 10$ and Figure C.2 for $L/D = 7$. Rows show variations in the sail appendage aspect ratio $b/c$, while columns show variations in hull-form cross-sectional aspect ratio $D_y/D_z$. In comparison to the ground-truth CFD simulations, the MSE is quantified at the top of each plot. Ground truth profiles can be viewed in Chapter 7. For all cases, the error is small. Other than minor anomalous distortions present around the sail wake, the overall flow fields are accurate.

Similar observations are made for the implicit-parameter network. These wake profiles are shown in Figure C.3 for $L/D = 10$ and Figure C.4 for $L/D = 7$. Network error, while small, is roughly double that of the explicit-parameter network. Anomalous distortions are also larger in size. This increase in error likely results from difficulties in the additional step of
learning vehicle geometric parameters implicitly from the vehicle geometries.

### 8.4.2 Performance in Inference

Network performance in inference is considered first. Figure 8.14 compares the generated wake flow field to CFD data for a case that was not included in the training data but still within the bounds of the data set. The inferred geometry is described by $D_y/D_z = 1.75$, $b/c = 0.45$, and $L/D = 8.5$. The left plot shows contours of the flow–field axial velocity defect for the generated wake profile, the center plot shows the CFD ground truth, and the right plot shows the associated error between the two profiles. The MSE, which is computed and labeled at the top of the figure, is shown to be small.

![Figure 8.14: Explicit–parameter CNN performance in inference.](image)

The implicit–parameter network shows poorer performance. Figure 8.15 compares the cGAN–generated wake profile to the ground–truth CFD for the $D_y/D_z = 1.75$, $b/c = 0.45$, and $L/D = 8.5$ geometry. These velocity defect profiles resemble the ground truth, however spurious flow features are present. The wake deficit of the sail is too large and some regions around the edge of the wake are anomalous. The computed MSE is larger than that of the explicit–parameter case. Error in the sail wake likely results from sensitivities in the method of inputting the geometry. Volume fractions computed near the sail are relatively small compared to those computed near the hull. As a result the network has difficulty accounting for sensitivities in sail changes.
Figure 8.15: Implicit–parameter cGAN performance in inference.

### 8.4.3 Performance in Prediction

Network performance in prediction is considered next. Figure 6.11 examines the generated wake profile for the \( \frac{D_y}{D_z} = 2.25 \), \( b/c = 0.55 \), and \( L/D = 11.0 \) geometry. This profile is compared against the ground–truth profile that was excluded from the training data. Error is larger for this case than for the inference case, likely because of the inherent difficulty of extrapolating results. Minor distortions appear around the sail wake. Error is still small, however, and the generated wake profile is qualitatively and quantitatively similar to the ground truth.

Figure 6.11 examines predictive performance of the implicit–parameter network for the \( \frac{D_y}{D_z} = 2.25 \), \( b/c = 0.55 \), and \( L/D = 11.0 \) geometry. Although capturing the overall wake features, the implicit–parameter network shows some anomalous distortions. MSE is similar to that of the explicit–parameter network. Large distortions are present outside of the primary wake region and the sail wake is, again, not perfectly captured. Distortions appear in the inner region of the wake that likely arise from difficulties in predicting results outside of the data set.

### 8.4.4 Performance for an Unconventional Geometry

The unconventional geometry is considered last and represents a case outside of the scope of the training data. The overall geometry and associated volume fractions were shown previously in Figure 8.9. This geometry was built with an asymmetric cross–section having \( D_y/D_z = 1.0 \) on the port side and \( D_y/D_z = 2.0 \) on the starboard side. The sail and slenderness are described by \( b/c = 0.5 \) and \( L/D = 10 \). Altogether, these features cannot be
Figure 8.16: Explicit–parameter CNN performance in prediction.

Figure 8.17: Implicit–parameter cGAN performance in prediction.
expressed to the explicit network with the three parameters alone. A new explicit network would have to be formulated with a parameter expressing cross-sectional asymmetry.

Figure 8.18 examines network performance for this unconventional geometry. Error is an order-of-magnitude larger for this case than for the inference case. Overall, the generated wake does not accurately capture the flow features of the true wake profile. Still, the generated profile is moderately successful in identifying the drag region on the right side of the wake as well as some of the thrust regions on the bottom and left side. The cGAN also attempts to generate the sail wake but has similar difficulties to the inference and prediction cases. The results suggest that the cGAN may be capable of mapping the parameter space for unconventional geometries given more data to properly formulate the latent parameter space.

![Figure 8.18: Implicit-parameter cGAN performance for an unconventional geometry outside of the scope of the training data.](image)

8.4.5 Performance from the Perspective of Integrated Kinetic Energy

Network performance is also viewed from the perspective of an integrated flow quantity, in this case the integrated axial kinetic energy \( KE_x \). The velocity defect is integrated over the domain of area \( A \) through the following equation.

\[
KE_x = \iint_A \frac{1}{2} \rho U_x^2 \, dA
\]  

(8.8)

In this equation, \( \rho \) is the density field. Kinetic energy is non-dimensionalized as \( 10^6 KE_x/(\rho_0 D^2 U_0^2) \). Table 8.4 shows \( 10^6 KE_x/(\rho_0 D^2 U_0^2) \) for both networks in the evaluation cases of inference.
and prediction. Performance is also shown for the unconventional geometry case using the implicit–parameter network. Kinetic energies of the network–generated profiles are compared to energies of the ground–truth profiles and the associated error is listed. The explicit–parameter network shows the best performance overall, showing small error in inference and prediction. The network well–characterizes the explicit parameter space performing better in inference than prediction. The implicit–parameter network, on the contrary, exhibits greater error in inference but smaller error in prediction. Larger overall error likely results from the difficulty of learning the underlying parameters of the input geometries in addition to learning the mapping between a geometry and its associated flow profile. The error for the kinetic energy of the unconventional geometry case is large, further showing the need to expand the training data set to consider these cases. Even with an expanded data set, a clear challenge is assessing the uncertainties of the network, especially for cases beyond the scope of the training data.

Table 8.4: Network performance using the integrated kinetic energy $10^6 KE_x/(\rho_0 D^2 U_0^2)$.

<table>
<thead>
<tr>
<th>Network type</th>
<th>Evaluation case</th>
<th>Generated</th>
<th>Ground truth (CFD)</th>
<th>Percent error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Explicit–parameter</td>
<td>Inference</td>
<td>90.3</td>
<td>89.4</td>
<td>0.9</td>
</tr>
<tr>
<td>Explicit–parameter</td>
<td>Prediction</td>
<td>239.3</td>
<td>228.7</td>
<td>4.6</td>
</tr>
<tr>
<td>Implicit–parameter</td>
<td>Inference</td>
<td>106.4</td>
<td>89.4</td>
<td>19.0</td>
</tr>
<tr>
<td>Implicit–parameter</td>
<td>Prediction</td>
<td>223.0</td>
<td>228.7</td>
<td>2.5</td>
</tr>
<tr>
<td>Implicit–parameter</td>
<td>Unconventional</td>
<td>226.0</td>
<td>139.8</td>
<td>61.7</td>
</tr>
</tbody>
</table>

8.5 Conclusions

In this study, two deep–learning CNNs were applied to the problem of generating wake profiles of a self–propelled marine vehicle. Using a database of 40 wake profiles extracted from computationally expensive CFD simulations, this data–driven approach was successful in creating a function to generate a realistic flow field in a fraction of a second. The first network was explicitly supplied parameters that described the marine vehicle hull–form geometry. The second network was supplied a tensor of volume fractions of the vehicle so that latent geometric parameters could be implicitly learned. The explicit–parameter network performed well in training, inference, and prediction, however it was limited to geometries described by the 3 prescribed geometric parameters. The implicit–parameter network was formulated to break this paradigm and consider any arbitrary geometry, including those that could not be described by the 3 parameters. Although performing reasonably in training, inference, and prediction, its performance was poorer than the explicit–parameter network. It showed particular difficulty in captured sail appendage effects due to sensitivities in the volume fraction tensors. The implicit–parameter network was capable of generating a wake profile for an unconventional geometry, showing some realistic flow features, however, the overall
result was erroneous. Although this wake profile was not acceptably accurate, its qualitative successes showed that the cGAN may be capable of accurately predicting the wake flow fields of unconventional geometry given a larger training data set.
Bibliography


Appendix C  Generated Training Profiles

Generated flow fields of axial velocity $U_x/U_0 - 1$ are presented for the explicit–parameter and implicit–parameter networks for 32 of the 40 training cases. The $b/c = 0.125$ cases were omitted for visual clarity. Figures C.1 and C.2 show contour maps of $U_x/U_0 - 1$ generated by the explicit–parameter CNN for $L/D = 10$ and $L/D = 7$, respectively. Figures C.3 and C.4 show contour maps of $U_x/U_0 - 1$ generated by the implicit–parameter CNN for $L/D = 10$ and $L/D = 7$, respectively.
Figure C.1: Explicit–parameter CNN generated training profiles and error for $L/D = 10$ geometries.
Figure C.2: Explicit-parameter CNN generated training profiles and error for \( L/D = 7 \) geometries.
Figure C.3: Implicit-parameter cGAN–generated training profiles and error for $L/D = 10$ geometries.
Figure C.4: Implicit–parameter cGAN–generated training profiles and error for $L/D = 7$ geometries.
Evaluation of the near–field flow of a marine vehicle can establish the vehicle’s efficiency and performance. The complete physics–based analysis, however, requires an expensive Computational Fluid Dynamics (CFD) simulation that can take days to compute. This process is prohibitively expensive for rapid evaluation, such as during the conceptual design phase where numerous concepts are considered. The present work accelerates the genesis of a design concept’s flow field from a process that takes several days to one of a fraction of a second by leveraging machine learning. A survey of marine vehicles are studied to establish the influence of various characteristics on the flow field. Over 50 CFD simulations were conducted in total, each with typical mesh sizes of 20–30 million cells and run times of up to 4 days using 512 cores in parallel. Two deep–learning networks were trained from these CFD simulations to create a functional mapping between vehicle geometry and the corresponding flow–field wake profile. The first network takes explicitly–specified geometric parameters as an input to generate the corresponding flow field. The second network takes discrete volume fractions of the vehicle geometry itself as input, allowing the network to implicitly learn the relevant design parameters. This implicit–parameter network shows the capacity to learn the design space of flow–field wake profiles for a large variety of input geometries not constrained to the explicit parameters, however a larger data set is required to achieve acceptable performance for unconventional geometries. The primary conclusions drawn from this work are listed.

1. Near–field flow develops independently of internal Froude number, but Reynolds number has an effect.

2. Mixing from a standard propeller generates a large amount of potential energy, while contra–rotating propellers effectively eliminate it.

3. Hull–form variations have a moderate influence on potential and kinetic energies.

4. Machine learning can be used to generate realistic flow fields for arbitrary input ge-
ometries, however expansion of the parameter space requires a corresponding increase in offline training data.

5. Hull–form geometric parameters can be implicitly learned by a machine learning algorithm to generate appropriate flow-field wake profiles.

6. An implicit–parameter approach can generate flow-field wake profiles for geometries even outside of the scope of the training data, although with greater error.

The entire process of building a ground–truth database and training a machine–learning model reveals the fundamental challenges of this overall approach. Finding adequate experimental data to validate the computational model is difficult, especially when no experiments exist for the desired flow regime. Given that the database is the ground truth to the deep–learning network, all computational uncertainties are propagated into the network–generated results in addition to uncertainties derived from the network parameter space characterization. The trained network, at best, matches the accuracy of the physics–based simulations. With the intended use of this overall approach being within the conceptual design phase, large uncertainties could cause costly issues later in the design process. It is desirable to have proper validation experiments to guide the computational formulation so as to simulate accurate ground–truth data and train an accurate network. Set–based concepts can also alleviate issues due to uncertainty.

Given these challenges, several avenues exist for potential future work. A detailed validation experiment of a marine vehicle can help guide the computational approach. This experiment should provide phase–averaged wake profiles in a stratified environment for a selection of vehicle concepts. The present simulations were conducted by solving the Reynolds–Averaged Navier Stokes equations with a two–equation turbulence closure model. As recognized by the literature review in Chapter 2, higher fidelity methods exist to better–model turbulence in the flow, although at a greater computational cost. While wall–resolved large eddy simulation (LES) is too expensive to make a broad data set of simulations in the near future, hybrid RANS/LES approaches may be feasible with the appropriate computational power. However, the greater computational expense will limit the size of the desired CFD data set used in machine learning. Increasing the variability of geometries and size of the data set should be pursued to better–generalize the deep–learning model to arbitrary and unconventional inputs. Furthermore, research should be conducted into establishing the breadth, resolution, and size of the ground–truth database required to accurately predict flow fields, especially for unconventional geometries beyond the scope of the training data. This work should also determine the level of accuracy that is acceptable for the conceptual design phase. Finally, the overall approach should be applied to the end–to–end design of a marine vehicle and compared to more traditional approaches. Possibilities for future work are listed.

2. Expand the ground-truth training CFD data set to include a broader range of geometries; for example, geometries could include stern appendages with various sizes at various locations.

3. Upgrade the simulation fidelity by applying a hybrid RANS/LES approach to near-field simulations; concurrently investigate the mechanisms of wake breakdown for the numerous self-propelled vehicle concepts in the improved ground-truth data set.

4. Within the context of hydrodynamics, physics-based simulation, and engineering design, establish rigorous performance metrics for the vehicle flow fields.

5. Determine the precise relationship between breadth, resolution, and size of the ground-truth data set and the accuracy of the predicted flow fields; further-determine the level of accuracy required for conceptual design.

6. Within the larger context of rapid evaluation in multidisciplinary design, apply these concepts to the successful end-to-end design of a marine vehicle, comparing to conventional approaches.