COMPUTATIONAL TECHNIQUES AND ANALYSIS OF CAVITATING-FLUID FLOWS

A Dissertation in
Aerospace Engineering

by

Michael P. Kinzel

© 2008 Michael P. Kinzel

Submitted in Partial Fulfillment of the Requirements for the Degree of

Doctor of Philosophy

August 2008
The thesis of Michael P. Kinzel was reviewed and approved* by the following:

Robert F. Kunz
Associate Professor of Aerospace Engineering
Thesis Advisor, Chair of Committee

Mark D. Maughmer
Professor of Aerospace Engineering

Phillip J. Morris
Boeing/A. D. Welliver Professor of Aerospace Engineering

David F. Hill
Associate Professor of Civil & Environmental Engineering

Jules W. Lindau
Research Associate
The Pennsylvania State University – Applied Research Laboratory
Special Member

Ralph W. Noack
Research Associate
The Pennsylvania State University – Applied Research Laboratory
Special Member

George A. Lesieutre
Professor of Aerospace Engineering
Head of the Department of Aerospace Engineering

*Signatures are on file in the Graduate School
ABSTRACT

A new numerical approach is presented and validated for valid for compressible-cavitating-fluid flows. The method employs modern interface capturing and computational mesh methods that improve the accuracy and enable a complex simulation capability. Specifically, the overset-mesh method is applied to facilitate complex and dynamic-multibody cavitating-vehicle simulations. In addition, a conforming, adaptive-mesh method is explored for a more efficient free-surface prediction capability. A numerical-sharpening scheme based on the level-set method is developed; the approach is valid for multiphase-compressible flows, accommodates phase change, and implements an efficient reinitialization scheme that is better suited for cavitating-fluid flows. The numerical methods are validated over a wide range of multiphase flows that include vaporous cavitation in water and cryogenic liquids, artificial cavitation, cavitating-body free-surface interactions, and dynamic-body multiphase simulations. The numerical schemes developed in this thesis are shown to improve the computational efficiency, the accuracy of the predicted flows, and the simulation capability for real-world multiphase computational-fluid-dynamics analysis.

Using the developed tools, a number of physical aspects of cavitating-fluid flows are investigated. The modeling approaches for partial cavitation simulations are evaluated and lead to conclusions regarding physical modeling requirements for accurate predictions. Finally, the physical behavior of cavitating flows is examined. Insight into cavity-air entrainment mechanisms, cavitating-oscillating hydrofoils, and cavitating-bodies near a free surface are obtained based on the validated methods and models.
# TABLE OF CONTENTS

LIST OF FIGURES .................................................................................................................. viii
LIST OF TABLES .................................................................................................................... xxi
NOMENCLATURE ................................................................................................................... xxii
ACKNOWLEDGMENTS .......................................................................................................... xxv

Chapter 1  Introduction and Background................................................................................ 1

1.1 Motivation....................................................................................................................... 1
1.2 An Introduction to Cavitation....................................................................................... 2
  1.2.1 Vaporous Cavitation............................................................................................. 3
  1.2.2 Artificial Cavitation.............................................................................................. 5
  1.2.3 Cavity-Free-Surface Interactions......................................................................... 10
  1.2.4 Other Applications............................................................................................... 11
  1.2.5 Physical Features of Cavitating Flows ............................................................... 11
1.3 Background of Interfacial Fluid Flow Modeling......................................................... 14
  1.3.1 Review of Navier-Stokes Based Methods............................................................ 15
    1.3.1.1 Surface Methods............................................................................................ 16
    1.3.1.2 Volume Methods ......................................................................................... 23
    1.3.1.3 Combined Methods ..................................................................................... 34
    1.3.1.4 Grid Methodologies ................................................................................... 35
    1.3.1.5 Turbulence Modeling .................................................................................. 39
  1.3.2 Review of Approximated Navier-Stokes Methods.............................................. 40
  1.3.3 Empirical Methods................................................................................................. 42
  1.3.4 Hybrid Methods..................................................................................................... 45
  1.3.5 Summary of Reviewed Computational Methods ............................................ 46
1.4 Thesis Scope and Outline .............................................................................................. 47
  1.4.1 Scope of Thesis..................................................................................................... 47
  1.4.2 Outline of Thesis .................................................................................................. 49
  1.4.3 Contributions........................................................................................................ 51

Chapter 2  Theoretical Model Formulation ............................................................................. 52

2.1 Governing Equations ..................................................................................................... 52
  2.1.1 Multiphase Flow Modeling .................................................................................. 53
  2.1.2 Turbulence Modeling ......................................................................................... 59
  2.1.3 Cavitation Modeling............................................................................................ 61
  2.1.4 Homogenous Mixing Rules ................................................................................ 62
2.2 Numerical Description .................................................................................................. 64
  2.2.1 Preconditioning Approach .................................................................................. 65
  2.2.2 Numerical Solution Procedure .......................................................................... 67
2.2.3 Advancement of Baseline ................................................................. 72
2.2.4 Computational Grid Generation ...................................................... 74
2.3 Summary of Modeling Approach .......................................................... 75

Chapter 3 Validation Case Description .................................................... 76

3.1 Test Cases ............................................................................................ 76
3.2 Artificial Cavitation Validation Cases ................................................... 76
  3.2.1 Test Case AC1: Disk Cavitator .......................................................... 77
  3.2.2 Test Case AC2: Artificially-Supercavitating Cone Cavitator .......... 78
  3.2.3 Test Case AC3: Artificially-Supercavitating Body ......................... 79
3.3 Vaporous Cavitation Validation Cases ................................................. 81
  3.3.1 Test Case VC1: Twisted Hydrofoil .................................................... 81
  3.3.2 Test Case VC2: Cryogenic Cavitating Hydrofoil ......................... 84
  3.3.3 Test Case VC3: Oscillating, Cavitating Wing ......................... 86
3.4 Free-Surface Validation Cases .............................................................. 89
  3.4.1 Test Case FS1: Submerged Hydrofoil .............................................. 89
  3.4.2 Test Case FS2: Surface-Piercing Hydrofoil ................................. 90
3.5 Summary of Test Cases ....................................................................... 93

Chapter 4 Accuracy Assessments and Validation Efforts ......................... 94

4.1 Assessment of Numerical Schemes ...................................................... 94
  4.1.1 Numerical Scheme Evaluation ....................................................... 95
  4.1.2 Preconditioning ............................................................................. 98
4.2 Grid Studies ......................................................................................... 101
  4.2.1 Grid Study Series 1: Artificial Cavitation ..................................... 102
  4.2.2 Grid Study Series 2: Natural Cavitation ....................................... 104
  4.2.3 Grid Study Series 3: Free-Surface Waves .................................... 109
  4.2.4 Grid Study Series 4: Multiphase Turbulence Simulations ............. 112
4.3 Time Resolution Requirements ........................................................... 113
4.4 Validation of Methods ......................................................................... 114
  4.4.1 Artificially Cavitating Flow Validation ...................................... 115
  4.4.2 Naturally Cavitating Flow Validation ....................................... 117
  4.4.3 Free-Surface Flow Validation .................................................. 129
4.5 Summary of the Investigations .............................................................. 134

Chapter 5 Numerical Method Enhancements for Developed Cavitation and
Discrete Interface Simulations ................................................................. 137

5.1 Overset-Mesh Approach for Multiphase Flows .................................. 137
  5.1.1 Description of Implementation .................................................. 138
  5.1.2 Examination of Overset Application for Multiphase Flows .......... 142
  5.1.3 Overset Mesh Strategies .............................................................. 150
    5.1.3.1 Cavitation Meshing Strategies ............................................... 151
    5.1.3.2 Free-Surface Meshing Strategies ........................................... 152
5.1.4 Validation of Overset-Mesh Implementation into UNCLE-M ..........153
  5.1.4.1 Dynamic-Mesh Validation: Oscillating Wing in a Water
          Tunnel .............................................................................................154
5.1.5 Dynamic Validation: Oscillating Hydrofoil Submerged Beneath a
          Free Surface......................................................................................155
5.1.6 Summary of Overset Approach Implementation..............................156
5.2 Mesh Adaptation for Free Surfaces ......................................................157
  5.2.1 Description of Method.................................................................158
  5.2.2 Submerged Hydrofoil.......................................................................161
  5.2.3 Surface Piercing Hydrofoil..............................................................163
  5.2.4 Summary of the Approach..............................................................170
5.3 A New Hybrid Volume-Fraction Based Level-Set Approach...............170
  5.3.1 Comparison of the Level Set and Volume Conservation
          Approaches.........................................................................................171
  5.3.2 Methods of Reinitializing the Interface............................................179
  5.3.3 Validation of the Method.................................................................182
  5.3.4 Summary of the Developed Level-Set Approach.............................190
5.4 Summary of Numerical Method Improvements .....................................191

Chapter 6 Improved Physical Models for Cavitation Modeling....................192
  6.1 Hybrid RANS/LES Turbulence Modeling for Cavitating Flows ..........192
        6.1.1 Artificial Supercavitation .........................................................195
        6.1.2 Natural Cavitation .....................................................................202
        6.1.3 Oscillating, Cavitating Wing......................................................211
        6.1.4 Summary of Improved Turbulence Models.................................219
  6.2 Computational Evaluation of Compressible Effects in Cavitating Flows ...220
        6.2.1 Predicted Effects of Compressible Fluids ..................................222
        6.2.1.1 Partial Cavitation: Twisted Hydrofoil ....................................222
        6.2.1.2 Partial Cavitation: Oscillating Hydrofoil ................................226
        6.2.2 Summary of the Compressible-Flow Modeling Effects .............233
  6.3 Summary..............................................................................................234

Chapter 7 Computational-Based Investigations of Physical Behavior .............235
  7.1 Numerical Examination of Cavitating Flows ......................................235
        7.1.1 Hysteretic Cavity Prediction.........................................................235
        7.1.2 Mechanisms of Air Entrainment from a Ventilated Cavity ..........241
        7.1.3 Summary of Supercavitating Flow Insights ...............................251
  7.2 Interactions of Cavitation with a Free Surface ....................................252
        7.2.1 Flat-Plate Predictions Compared to Boundary Element Theory ....252
        7.2.2 Three-Dimensional Predictions .................................................254
        7.2.3 Ventilated Cavitation .................................................................259
  7.3 Investigations of Oscillating, Multiphase Lifting Surfaces ....................267
        7.3.1 Oscillating a Hydrofoil Submerged Beneath a Free Surface .......267
LIST OF FIGURES

Figure 1-1: Vortex, sheet, and cloud cavitation examples occurring on a lifting surface. This wing is operating at $\sigma_v=0.43$ and is pitched 9.5°. The photograph is borrowed from Brennen [2]. ..........................................................5

Figure 1-2: A glassy supercavity that forms behind a conical-shaped cavitator and enshrouds the silver-colored body. The photograph is borrowed from Stinebring et al. [5]. .................................................. 6

Figure 1-3: Typical ventilation rate relations. Solid lines represent the $C_Q-\sigma_c$ relation at a given $Fr_D$. Dashed lines represent the constant $\sigma Fr$ values. Diagrams of each closure type are displayed to the right. The cavity-shape diagrams are borrowed from Semenenko [4]. ......................................................7

Figure 1-4: Sound speed as a function of air-volume fraction in air/water, vapor/water mixtures and when a vaporization process occurs............................13

Figure 1-5: Overset grid used to model a dynamic bubble. Figure borrowed from Fast [17]. ........................................................................................................17

Figure 1-6: Level-set-based LES of a plunging-breaking wave compared to experiment. Simulations shown to the right with contour plots of vorticity and black lines indicate the predicted interface. (a) and (b) indicate different instances in time. All figures are borrowed from Yang and Stern [21]................23

Figure 1-7: Sample cell volume-fraction distribution with various interface reconstruction methods. The line represents true interface, shaded regions are the reconstructed interface computed with (a) SLIC and (b) PLIC schemes. Diagrams borrowed from Scardovelli and Zaleski [41]. ......................................32

Figure 1-8: Example of the advection step in the VOF method using a PLIC interface reconstruction. Here the interface advects in the x-direction from $ab$ to $cd$. Figure borrowed from Scardovelli and Zaleski. [41].............................33

Figure 1-9: Description of a quadtree refinement strategy. ........................................36

Figure 1-10: Example of adaptive-mesh refinement applied to a breaking wave. Figure borrowed from Hay el al. [38]. .................................................................37

Figure 1-11: Computations of free-surface predictions using the 2D+t methodology. Each slice is a prediction at another instance in time, which is modeled to represent the stream-wise direction. Figure borrowed from Andrillon and Alessandrini [48]. .................................................................41
Figure 1-12: Domain decomposition method, a level-set based CFD approach is used in the region $\Omega^F$, and a potential flow is used in $\Omega^B$. Coupling of the domains is made at $\Gamma$. Diagram borrowed from Iafrati and Campana [51]........45

Figure 2-1: Contour plot of the drag to buoyant force ratio, $F_{D/B}$, (on a log scale) that a bubble experiences over various flow conditions. High values indicate a valid homogenous velocity assumption. The low values indicate that a strong buoyancy-driven slip velocity may exist..................................................58

Figure 3-1: Axisymmetric disk cavitator (6° wedge simulated) ventilated on the aft faces of the cavitator. The conditions are $Re_N=90\times10^6$, $Fr_N=\infty$, $C_Q=0.5$. The isometric view shown in part (a) has a cavitator colored by pressure and the cavity shape ($\alpha=0.5$ isosurface) shown in blue. The side view and the medium-resolution computational mesh are shown in part (b), where the contours are colored by the volume fraction (red=air, blue = water)..............................77

Figure 3-2: Mesh used for the 15°-half-angle cone cavitator. The mesh in the upper part displays the entire domain. In the lower part, a detail of the cavitator (black), pseudo body (gray), and the ventilation port (yellow) are all shown.. ....................................................................................................................79

Figure 3-3: Experimental setup description of the geometry (a) and a photograph displaying cavity, body, and support strut. These figures are borrowed from Schauer [72]..........................................................................................................80

Figure 3-4: The computational meshes used for simulations of the supercavitating body. ..............................................................................................................................................81

Figure 3-5: Twisted wing (a) geometry, (b) modeled geometry, and (c-d) mesh used in these investigations. .................................................................................................................83

Figure 3-6: Photograph of cavitating hydrofoil case (in hydrogen) and description of the experiment. The photograph is borrowed from Hord [75]..................................85

Figure 3-7: Computational mesh overlaid on a photograph of the experiment [75]. ..86

Figure 3-8: Experimental measurements of the lift [77] for a pitching hydrofoil with varied cavitation numbers.................................................................87

Figure 3-9: General overset-grid scheme used for the three-dimensional cases. The simulated walls are colored by pressure, and slices at the mid-span plane and through the trailing edge are shown in red and blue respectively. The remained tunnel walls are modeled with displacement thickness.........................88

Figure 3-10: Overset grid system used to examine free-surface mesh requirements. The red grids remain static, while the blue (free-surface
resolving) and black (water-field resolving) meshes are refined and coarsened. The black line indicates the free-surface location.

Figure 3-11: Description of experiments of a surface-piercing, NACA 0024 hydrofoil. Both the diagram and photograph are borrowed from Metcalf et al. [79].

Figure 3-12: Computational mesh used for the surface-piercing, NACA 0024 hydrofoil. Upper plot shows the entire domain, flow goes from right to left. In the lower plots, the flow goes from left to right, and in the left and right plots are the side and top views, respectively.

Figure 4-1: Predicted volume-fraction profiles through a cavity interface using various numerical schemes. The comparison displays the interface-sharpness preserving ability. This is for the axis-symmetric-disk cavitator (Case AC1) at $Re_N=90\times10^6$, $Fr_N=\infty$, $C_Q=0.5$.

Figure 4-2: Comparison of the semi-empirical and CFD cavity-profile predictions using various numerical schemes. Part (a) is the predicted cavity profile, given by the $\alpha=0.5$ isosurface. The error from the semi-empirical solution in part (b) is based on a $\sigma_i$ from the Roe-prediction, and in part (c) a prediction-specific $\sigma_c$ value is used. This is for the axis-symmetric-disk cavitator (Case AC1) at $Re_N=90\times10^6$, $Fr_N=\infty$, $C_Q=0.5$. Note that the axial distance in the x-axis is implied for all figures.

Figure 4-6: Comparison of the predicted, time-averaged, pressure distributions at the midspan of the twisted hydrofoil using various mesh resolutions. This comparison is for $\sigma=1.07$ (Case VC1) and using a DES approach.

Figure 4-7: Sensitivity of mesh resolution on wave predictions (a-c) and the loads on the submerged body (d-f). The blue ‘x’ indicates mesh refinement only in the free-surface normal direction, whereas the black ‘o’ represent mesh refinement only in the axial direction.

Figure 4-8: Stream-wise, non-dimensional, pressure gradient $(c/q_\infty dp/dx)$ due to developed wave train.

Figure 4-9: Pressure spectra at various grid resolutions. The pink line is the theoretical value for isotropic turbulence, $(fc/U)^{7/3}$. With mesh refinement, the simulation of additional inertial-range turbulent eddies are apparent and the behavior converges to the theoretical behavior.

Figure 4-10: Compared of CFD predictions to experiment for a supercavity forming around a 15°-half-angle conical cavitator (Case AC2). The observed cavity (a) is quite similar to that predicted (b) at a similar $\sigma_c$ value. The
predicted cavitator loads (c) compare well with correlations to measurements
developed by May [13]........................................................................................................116

Figure 4-11: Comparison of the cavity predictions to observed cavities from
experiments [72] at similar cavitation numbers. The CFD-prediction
geometry is colored by the pressure, and the blue, black, and grey isosurfaces
are at vapor-volume fractions of 0.97, 0.8, and 0.05, respectively. The
photographs are borrowed from Schauer [72]..............................................................117

Figure 4-12: Sequence of CFD solutions (left), from view of the suction surface,
through nondimensional time ($t^*=tU/c$) using isothermal-compressible DES
assumptions for the twisted wing at $\sigma_v=1.07$ (Case VC1). The blue and grey
translucent isosurface are at vapor-volume fractions of 0.5 and 0.1,
respectively. At the right are photographs borrowed from experiments of
Foeth et al. [73]...........................................................................................................121

Figure 4-14: Predicted pressure distributions. Pressure contours on the suction
surface are given in part (a). Comparisons of the CFD predicted pressure to
data reported [83] at various span-wise locations, are provided in part (b) and
a corrected form in part (c). Note that the corrected form assumes an error in
the velocity used for the reported, nonphysical, $c_p$ values......................................124

Figure 4-15: Time-averaged solutions of a cavitating hydrofoil in a cryogenic
(liquid-nitrogen) flow (Case VC3), at $Re_c=9.1\times10^6$, $T_\infty=83.06K$, and $\sigma_\infty=1.7$.127

Figure 4-16: Time-averaged solutions of a cavitating hydrofoil in a cryogenic
(liquid-nitrogen) flow (Case VC3), at $Re_c=9.1\times10^6$, $T_\infty=83.06K$, and $\sigma_\infty=1.7$.
The surface profiles for the pressure coefficient, temperature, vapor-volume
fraction, and the standard deviations (of those measures) are provided...............128

Figure 4-17: Comparison of the experimental measurements [78] and the
predicted free-surface-wave profiles formed by a submerged NACA 0012
hydrofoil (Case FS1) at $Re_c=1.45\times10^5$, $Fr_c=0.57$, and $\alpha=5^\circ$. In part (a), the
foil is submerged 1.29$c$ below the nominal free surface and creates a
nonbreaking wave, and in part (b), a breaking wave is formed when the foil is
submerged 0.783$c$ below the nominal free surface. .....................................................130

Figure 4-18: Comparison of the predicted and measured [79] free-surface
elevations, relative to the nominal elevation. Note that to experimental
measurements are in the box to the lower left. .............................................................132

Figure 4-19: In part (a), the CFD predictions are compared to the experimental
measurements [79] of the water line. A photograph from the experiment, near
the foil surface, is provided in part (b). .................................................................133
Figure 4-20: Predicted pressure coefficient distribution compared to the experimental measurements. The white-dashed line in part (b) outlines the range where the pressure was measured on the wing. ..................................................134

Figure 5-1: Sample overset mesh for a business jet. The black nodes, or fringe points, are located at the domain boundaries of the green mesh and interpolate values from the cyan mesh. The red nodes, or fringe points, make an intergrid domain boundary for the cyan mesh and interpolate values from the green mesh. The cyan mesh has “out”cells within, and in the proximity of, the jet that are not shown. This figure borrowed from the PEGASUS website [84]..........................................................................................................139

Figure 5-2: Example of behavior of the volume fraction, $\alpha$, and the smoothed volume fraction, $\alpha'$, in the proximity of an interface. Note that $\alpha'$ approaches a more linear function near the interface. ..............................................................142

Figure 5-3: Grids and cavity interface predictions (shown in black/white regions) from the overset-grid refinement studies on the axisymmetric disk cavitator at $Re_N=90\times10^6$, $Fr_N=\infty$, $C_Q=0.5$ (Case AC1). The baseline mesh is displayed in red, and the blue mesh is the overlapped mesh with various resolutions. ...............144

Figure 5-4: Effect of overset-grid refinement on the predictions compared to semi-empirical theory for an axisymmetric disk cavitator at $Re_N=90\times10^6$, $Fr_N=\infty$, $C_Q=0.5$ (Case AC1). ........................................................................................................145

Figure 5-5: Effect of overset-grid alignment and interpolation method on the predictions, compared to semi-empirical theory for an axisymmetric disk cavitator at $Re_N=90\times10^6$, $Fr_N=\infty$, $C_Q=0.5$ (Case AC1)...............................................................147

Figure 5-6: Interpolation quality using various interpolation methods on a Cartesian–overlapped mesh. The white contours indicate the interpolated values and the black contours indicate the donor values. .........................................................148

Figure 5-7: Interpolation quality using various interpolation methods on a skewed–overlapped mesh. The white contours indicate the interpolated values and the black contours indicate the donor values. .........................................................150

Figure 5-8: Typical overset mesh for supercavitating vehicle simulations. ...............152

Figure 5-9: General grid scheme used for free-surface predictions..............................153

Figure 5-10: Comparisons to experiment [77] with corrections based on the effect of transitional flow on the lift. ........................................................................................................155
Figure 5-11: Comparison of experimental loads [87] from dynamic stall case compared to simulations of a submerged, oscillating hydrofoil at various depths....................................................................................................................156

Figure 5-12: Sample of the mesh adaptation to a free surface with breaking waves...160

Figure 5-13: Free-surface adaptation for a submerged NACA 0012 hydrofoil (Case FS1). ........................................................................................................................................162

Figure 5-14: Details of adaptive refinement on interface (a) volume fraction with smeared interface (b) length scale determined from turbulence model.......163

Figure 5-15: (a) Adapted grid to the free surface of the surface-piercing NACA 0024 hydrofoil (Case FS2). (b) The predicted free surface, shown by the isosurface at $\alpha=0.5$, aligns with the adapted mesh. The black gridlines indicate the adaptation on the surface of the hydrofoil. The red gridlines indicate a slice through the $x/c=0.95$ location.....................................................164

Figure 5-16: Comparison of adapting-mesh and nonadapting-mesh solutions to experimental measurements [79] of the free-surface elevation for the surface-piercing strut (Case FS2). ........................................................................166

Figure 5-17: Predicted pressure distribution, $c_{p,dyn}$, compared with the experiment of the NACA 0024 surface piercing strut (Case FS2). (a) Experiment measurements [79], compared to (b) baseline and (c) adaptive predictions........167

Figure 5-18: Side view of surface piercing NACA 0024 hydrofoil (a) prediction using adaptive method ($\alpha=0.9$ isosurface colored by elevation) (b) experimental photograph [79]........................................................................168

Figure 5-19: Quarter view of a surface piercing NACA 0024 hydrofoil (a) streamlines hydrofoil surface are colored by volume fraction (b) $\alpha=0.9$ isosurface colored by $1/|\nabla\alpha|$ to display mixture regions (c) photograph from experiments [79]. ..................................................................................................169

Figure 5-20: Comparison of the residual of the reinitialization equation using Eq. 5.22 (Signed Dist. Func.) and Eq. 5.25 (volume fraction based). These include analytic and numerical comparisons for various mesh densities and numerical schemes................................................................................................178

Figure 5-21: Example of the sharpening procedure for the realizable scaled-sharpening approach. ........................................................................................................181

Figure 5-22: Effect of the level-set formulations on interface-sharpness preserving for an axisymmetric disk cavitator (Case AC1). ......................................................182
Figure 5-23: Effect of the mesh refinement using level-set formulation 1, applied to an axisymmetric disk cavitator (Case AC1) ................................................................. 184

Figure 5-24: Effect of the mesh refinement using level-set formulations 2 and 3, applied to an axisymmetric disk cavitator (Case AC1). ..................................................... 185

Figure 5-25: Convergence of the estimated error in the (a) computed cavity profile and (b) the $C_{D,0}$ values with mesh refinement. The rate of convergence across the examined range is indicated ............................................................................. 187

Figure 5-26: Test case displaying the solution at 3 ms after a Mach 11.38 shock wave strikes a gas-water interface. ............................................................................... 189

Figure 6-1: Effect of turbulence model compared to data and photographs from experiments by Schauer [72]. ................................................................................. 196

Figure 6-2: Diagram of the air-boundary layers within a cavity, theorized as an important mechanism for air entrainment. Diagram borrowed from Spurk [89] ........................................................................................................................................ 199

Figure 6-3: Comparison of the predicted volume fraction field (in upper part of the sub figures), and the turbulent intensity (in the lower part of the subfigure) using a RANS and DES turbulence model.............................................. 200

Figure 6-4: Predicted cavity boundary layers for the axisymmetric disk cavitator (Case AC1) using laminar and RANS methods. This distance corresponds to the distance from the interface, into the air in the cavity. The velocity magnitude is used to account for the skewness of the cavity and the sign(U) isolates entrained air from recirculating air in the cavity. ........................................ 201

Figure 6-5: Time-averaged solution views of the twisted foil (Case VC2) using an URANS approach. The wing is colored by $c_{p,b}$, the red streamlines pass over the suction surface, the grey streamlines pass over the pressure surface, the blue translucent isosurface is at $\alpha=0.5$, and the grey isosurface is at $\alpha=0.1$, and the black lines are surface streamlines ................................................................................. 204

Figure 6-6: Time-averaged solution views of the twisted foil (Case VC2) using an unsteady DES approach. The wing is colored by $c_{p,b}$, the red streamlines pass over the suction surface, the grey streamlines pass over the pressure surface, the blue translucent isosurface is at $\alpha=0.5$, and the grey isosurface is at $\alpha=0.1$, and the black lines are surface streamlines ................................................................. 205

Figure 6-7: Time-averaged values of the DES switch, $F_{DES}$, for the twisted, cavitating wing (Case VC1) at various span-wise locations. The blue translucent isosurface is at $\alpha=0.5$, and grey at $\alpha=0.1$ .................................................. 206
Figure 6-8: Time-averaged solution views of the twisted foil (Case VC2) using a MILES approach. The wing is colored by $c_{p,b}$, the red streamlines pass over the suction surface, the grey streamlines pass over the pressure surface, the blue translucent isosurface is at $\alpha=0.5$, and the grey isosurface is at $\alpha=0.1$, and the black lines are surface streamlines...........................................................208

Figure 6-9: Comparison of the simulations using various turbulence model approaches to an experimental video [74] compressed into a single image. The cavity from the experiment is outlined by the red-dashed line and the mixture region is indicated using the yellow-dashed line. The dark-blue and light-blue isosurfaces are at $\alpha=0.7$ and $\alpha=0.3$, respectively.................................209

Figure 6-11: Comparisons of predictions (with turbulence model variations) and experiments during the secondary cavity cycle and the recovery from the cavitation event in a hydrofoil oscillation. The columns, from left to right, respectively display the cavities observed in experiments [77], URANS, DES, and MILES turbulence model predictions. The pink and white-translucent isosurfaces represent the $\alpha=0.5$ and 0.1 values, respectively. ..............................215

Figure 6-12: Comparison of the load and moment predictions using various turbulence modeling approaches for a cavitating, oscillating wing. The vertical-dashed lines represent the times where the solution is extracted in Figs. 6-10 and 6-11........................................................................................................217

Figure 6-13: Power spectra of the lift loads on the fin from predictions using various turbulence models and compared to experiments. The pitch cycle from the CFD simulation (CFD: pitch) displays the power spectra based on three-dimensional lifting-line theory and the simulated pitch cycle. Similarly, the experimental pitch cycle (Exp.: pitch) extends the pitch cycle from the experiments to a lift-power spectra curve.............................................................218

Figure 6-14: Time-averaged solution views of the twisted foil (Case VC2) using a DES, isothermal compressible approach. The wing is colored by $c_{p,b}$, the red streamlines pass over the suction surface, the grey streamlines pass over the pressure surface, the blue translucent isosurface is at $\alpha=0.5$, and the grey isosurface is at $\alpha=0.1$, and the black lines are surface streamlines. ......................223

Figure 6-15: Time-averaged solution views of the twisted foil (Case VC2) using a DES, fully compressible approach. The wing is colored by $c_{p,b}$, the red streamlines pass over the suction surface, the grey streamlines pass over the pressure surface, the blue translucent isosurface is at $\alpha=0.5$, and the grey isosurface is at $\alpha=0.1$, and the black lines are surface streamlines. ......................224

Figure 6-16: Comparison of the simulations using various state equation assumptions to an experimental video compressed into a single image. The
cavity from the experiment is outlined by the red-dashed line and the mixture region is indicated using the yellow-dashed line. The dark-blue and light-blue isosurfaces are at $\alpha=0.7$ and $\alpha=0.3$, respectively.

Figure 6-17: Comparisons of state equation assumptions and experiments during the initial cavity cycle. The columns, from left to right, respectively display the cavities observed in experiments [77], incompressible, isothermal compressible, and full energy conserving compressible with $p_v(T)$. All cases use the DES turbulence model. The pink and translucent white isosurfaces represent the $\alpha=0.5$ and 0.1 values, respectively.

Figure 6-18: Comparisons of state equation assumptions and experiments during the secondary cavity cycles and recovery. The columns, from left to right, respectively display the cavities observed in experiments [77], incompressible, isothermal compressible, and full energy conserving compressible with $p_v(T)$. All cases use the DES turbulence model. The pink and translucent white isosurfaces represent the $\alpha=0.5$ and 0.1 values, respectively.

Figure 6-19: Comparison of the load and moment predictions using various state-equation assumptions. Note that the vertical-dashed lines represent the times that the solution views of Figs. 6-17 and 6-18 are extracted from.

Figure 6-20: Power spectra of the lift loads on the fin from predictions using various state equation assumptions (with DES) and from the experiments. The pitch cycle from the CFD simulation (CFD: pitch) displays the power spectra based on three-dimensional lifting-line theory and the simulated pitch cycle. Similarly, the experimental pitch cycle (Exp.: pitch) extends the pitch cycle from the experiments to a lift-power spectra curve.

Figure 7-1: Predicted hysteretic cavity behavior compared to experiments [12] of a ventilated supercavitating body (Case AC3). (a) Comparisons of the experimentally obtained and computed $C_Q-\sigma_c$ curves revealing hysteresis and the (b) corresponding boundary layer air recovered, $k_{REC}$.

Figure 7-2: Predicted cavity shape, length, and corresponding ventilation rate through the hysteresis loop of the supercavitating body case (Case AC3). Note that the predicted cavities are displayed with the $\alpha=0.8$ isosurface, and the black, purple, blue, red and pink cavity shapes are for $C_Q=0.08$, 0.2, 0.32, 0.45, and 1.0, respectively.

Figure 7-3: Comparisons of predicted cavity shapes to experimental photographs of the supercavitating body (Case AC3). The photographs in the upper part of each subfigure are borrowed from the experiments of Schauer [72], and the
center part of each subfigure is the high-\(\sigma\) solution, and the lower part of each subfigure is the low-\(\sigma_c\) solution.

Figure 7-4: Examination of the experimental results of a supercavitating body (Case AC3) [72], and the RANS and URANS CFD predictions compared to the Spurk’s theory.

Figure 7-5: The \(C_Q=0.45\) RANS prediction of the ventilated, supercavitating body (Case AC3). The contours display the local air entrainment, \(Q_{LOC}\), and the streamlines indicate the path of the entrained versus cavity air. Note that little air escapes the cavity region and the predictions display air entrainment along cavity shear layers.

Figure 7-6: Air entrainment and recovery for twin vortex cavity on a cone cavitator (Case AC2). The streamlines (colored by pressure gradient) are traced backwards to indicate the origin. The contour lines display the local air entrainment rate, \(Q_{LOC}\). The background is a contour plot of the air volume fraction with black indicating air.

Figure 7-7: Comparison of the \(C_Q-\sigma_c\) curve for various theories to experiments of a hysteretic cavity attached to a disk cavitator [92] with a corresponding \(k_{REC}, V_{VORTEX}/V_{\infty}, V_{BL}\), and the \(V_{DP}\) curves.

Figure 7-8: Proposed form of modeling \(C_Q-\sigma_c\) relation.

Figure 7-9: Cavity and free-surface predictions occurring near a ventilated flat plate. The flat plate is positioned 1.0 \(c\) beneath the undisturbed free surface. The CFD predictions are compared to boundary element method predictions from Faltinsen and Semenov [93]. The upper line indicates the free surface, whereas the oval-shaped line represents the cavity interface. The dashed vertical line is a corrected cavity length from the theory [93] based on the CFD computed \(\sigma_c\).

Figure 7-10: Translucent view of the predicted free surface colored by elevation (top) and vertical velocity in the \(x-z\) plane passing through the trailing edge. Note that the free surface affects the wake on the tip vortex induced flow.

Figure 7-11: Surface wave elevations for a finite-span wing beneath a free surface. Comparisons are made between cases with a far field (black contours in upper half) and an inviscid wall (white contours in lower half) placed 3.0\(c\) past the tip.

Figure 7-12: Side by side comparison of the free surface prediction for a cavitating (\(\sigma_c=0.5\)) and noncavitating control fin, submerged 0.783\(c\) below the surface, at \(Fr_c = 9.23\). (a) The cavitating case is displayed with the blue isosurface representing the cavity, and is compared to the case with no cavity.
Comparisons of the predicted free-surface elevations. (c) The predicted reentrant jet forming in the cavitating simulations.  

Figure 7-13: Comparison of vaporous and ventilated cavitation predictions from a supercavitating hydrofoil. The conditions correspond to $Fr_c = 8.23$, $\alpha = 10^\circ$, $Re_c = 1.4 \times 10^6$. The ventilated case has a $\sigma_c = 0.04$ whereas the vaporous case uses $\sigma_v = 0.25$. In parts (a) and (b) the $\alpha = 0.5$ isosurface displays the cavity shape.  

Figure 7-14: Comparison of computed surface wave heights above fully wetted 4:1 spheroid (upper half) and naturally cavitating ($\sigma = 0.318$ at centerline depth) blunt nosed object (lower half) made of spheroidal sections. Each object is one-half body length (0.5L) beneath free surface, at centerline, with $Re_L = 2.97 \times 10^8$ and $Fr_L = 3.11$.  

Figure 7-16: Predictions over a 4:1 aspect ratio body submerged below a free surface at nominal centerline depth of 0.5L, $Re_L = 2.9 \times 10^8$, and $Fr_L = 3.12$. The displayed cases are the (a) baseline fully wetted and (b) a supercavitating body with a disk cavitator and cavity highlighted by the $\alpha = 0.5$ isosurface, visible through the translucent free surface of a liquid volume fraction equal to 0.5.  

Figure 7-17: Predictions over a 4:1 aspect ratio body submerged below a free surface at nominal centerline depth of 0.5L, $Re_L = 2.9 \times 10^8$, and $Fr_L = 3.12$. (a) The streamlines originate from the gas deflectors. The gas filled vortices are highlighted with 3 selected downstream contours colored by the gas volume fraction (red=1, blue=0). (b) View from beneath of the free-surface deflections. (c) View of the induced vortical flow interacting with the free surface colored by the gas-volume fraction (blue=water, red=gas).  

Figure 7-18: Effect of scaling time for the single-phase case, $\sigma_v = 4.0$.  

Figure 7-19: Effect of pitch frequency on the predicted loads of a two-dimensional simulation. Note that the $f_t = 1.0$, 0.5, and 0.1 load predictions are smoothed for clarity; the actual fluctuations are near those of the $f_t = 0.05$ case. The vertical-dashed lines correspond to the flow-field visualizations in Fig. 7-20.  

Figure 7-20: Comparison of the predicted flow field during the first cavity cycle for various pitch rates. Each row is at a similar normalized time, $t^*$, and $\alpha$, which can be referenced to the loads in Fig. 7-19. The scaling factors are indicated at the bottom of each column. The contour plots display the vapor-volume fraction (grey indicates the amount of vapor). The line plot displays the corresponding foil-surface $-c_p$ values. 
Figure 7-21: Various prediction methods of the finite wing hydrodynamics..............274

Figure 7-22: Comparison of two- and three-dimensional predictions at \( t=0.05 \). In each subfigure, the upper plot is a three-dimensional solution representation and the lower is from a two-dimensional prediction. The 3D case displays the surface streamlines (in red), vapor-volume fraction isosurfaces at 0.5 (pink) and 0.1 (white), and has solid surfaces colored by pressure. The surface pressures at three spanwise stations are also provided (red => \( z/c=0.25 \), yellow => \( z/c=1.0 \), and green => \( z/c=1.8 \)). As before, the two-dimensional predictions display the vapor-volume fraction in grey and the surface pressure distribution.................................................................276

Figure 7-23: Predicted events of a reentrant jet and cavity collapse. The surfaces are colored by the liquid-volume fraction. Vectors of the vaporeous regions are plotted at various spans along the wing. The black lines are surface-streamlines. ...........................................................................................................278

Figure A-1: Diagram of the change in the distance function with respect to \( x \) and \( y \). ...........................................................................................................................298

Figure B-1: Comparison of volume conservation of air for an unsteady flow at different time step sizes and values of \( \beta \). For more ideal \( \beta \) values, of 10, decreasing the time step size has no effect, at high \( \beta \) the volume conservation error is increased with smaller time steps.................................................................307

Figure B-2: Comparison of volume conservation of air for an unsteady flow for a varied number of iterations in pseudo time (IC) and different Courant number values (CFL). Here the ability of the scheme in preserving volume is highly dependent on the number of inner cycles taken, or the nonlinear convergence characteristics. ......................................................................................................308

Figure B-3: Comparison of integral parameters for a supercavitating disk cavitator at \( Fr_N=10.0 \) and \( Re_N=1\times10^6 \), with fully turbulent, transitional, and laminar modeling assumptions. ..................................................................................309

Figure B-4: Comparisons of the predicted cavity shape with various turbulence model assumptions. Conditions correspond to \( Fr_N=10.0 \) and \( Re_N=10^6 \), the cavities assumed to be the \( \alpha=0.5 \) isosurfaces, which is colored by \( 1/|\nabla \alpha| \). The \( C_Q \) values used are 1.5, 1.0, 0.5, 0.3, and 0.2 going from the upper to the lower cavities. ........................................................................................................310

Figure B-5: Effects of assuming a symmetry plane at the vehicle centerline..............311

Figure B-6: Comparison of a predicted cavity with and without a symmetry plane. The conditions for this case are: \( C_Q=1.5, Fr_N=10, Re_N=1\times10^6 \).............312
Figure B-7: Effect that the density of the ventilated gas has on integral cavity parameters for a disk cavitator at $Fr_N=10$, $Re_N=1\times10^6$ .................................................. 314

Figure B-8: Disk cavitator at $Fr_D=10$, $Re_D=1\times10^6$, at various ventilation rates and for different densities of the noncondensable gas injected. The ventilation rates correspond to $C_Q$ values of 1.5, 1.0, 0.5, 0.3, and 0.2 from top to bottom for each of the densities given. ................................................................. 315

Figure B-9: The predicted effect of modeling laminar flow for the investigated cross section ................................................................................................................. 317

Figure B-10: Various prediction methods of the finite wing hydrodynamics. ........ 319

Figure B-11: Comparisons to experiment with an estimated effect of transitional flow ......................................................................................................................... 320
LIST OF TABLES

Table 1-1: Overview of the approaches to handle interfaces...............................46
Table 3-1: Table summarizing the validation cases...........................................93
Table 4-1: Force prediction with respect to spatial resolution for a disk cavitator..103
Table 4-2: Description of various O-grids used for the hydrofoil....................105
Table 4-3: Summary of hydrofoil forces..........................................................106
Table 4-4: Effect of time step size on a marginally stable cavity......................114
Table 4-5: Predicted loads on the twisted wing compared to the measured lift...119
Table 6-1: Lift predictions compared to measurements with varied turbulence
models...........................................................................................................211
Table 6-2: Comparison of the predicted lift to the experiments.......................226
NOMENCLATURE

Symbols

\( a \) speed of sound

\( A \) cross-sectional area of duct

\( A_R \) wing planform area \((bc \text{ for rectangular planform})\)

\( C_{L2} \) empirical constant for cavity size-pressure relation

\( c_{p} \) cross-sectional area of a duct

\( c_l \) two-dimensional lift coefficient, \( L/(q_\infty c) \)

\( C_L \) three-dimensional lift coefficient, \( L/(q_\infty A) \)

\( c_d \) two-dimensional drag coefficient, \( D/(q_\infty c) \)

\( C_D \) three-dimensional drag coefficient, \( D/(q_\infty A) \)

\( c_p \) pressure coefficient, \( (p-p_\infty)/q_\infty \)

\( c_{p,dyn} \) specific heat at constant pressure

\( c_{p,dyn} \) with removed hydrostatic pressure, \( (p_\infty-(p-\rho gd))/q_\infty \)

\( c_v \) specific heat at constant volume

\( C_Q \) ventilation rate coefficient, \( QV_\infty^{-1} D^{-2} \)

\( d \) submergence depth

\( D \) drag

\( D_N \) cavitator diameter

\( e \) internal energy

\( Eu \) Euler number, \( 2p_\infty (rV_\infty^2)^{-1} \)

\( f_t \) pitch cycle time scaling factor

\( g \) gravity

\( h \) enthalpy

\( h_{lg} \) height of free-surface

\( h_{lg} \) latent heat of vaporization

\( H \) heavyside function

\( k \) specific turbulent kinetic energy

\( k \) reduced frequency, \( \alpha U^{-1/2} \)

\( k \) scaling constant

\( k \) wave number

\( k_Q \) empirical constant for cavity boundary layer entrainment rate

\( k_{REC} \) portion of boundary-layer entrained air recovered back into cavity

\( L \) body length

\( L \) reference length

\( n \) lift

\( n \) number of species

\( M \) Mach number, \( V/a \)

\( p \) pressure

\( Pr \) Prandtl number

\( q \) dynamic pressure, \( 1/2 \rho V^2 \)

\( q \) turbulent intensity, \( k^{0.5} \)
ventilation rate
$Q_{LOC}$ local air entrainment rate, $\alpha u/u_\infty$
$R_c$ cavity radius
$R_N$ cavitator radius
$Re$ Reynolds number, $\rho V L/\mu$
$s$ length along perimeter of object
time in semi-chords, $2U t/c$
time
$t^*$ nondimensional time, $t V/c$
nondimensional time in pitch cycle, $t/\text{period}$
$T$ temperature
$V$ velocity
$V_{BL}$ axial velocity in vortex tube based on boundary layer entrained gas
$V_{DP}$ axial velocity in vortex tube driven by pressure gradients
$V_{VORTEX}$ axial velocity in vortex tube
$We$ Weber number, $\rho V^2 L/\sigma_{ST}$
$Y$ mass fraction

Greek Symbols
$\alpha$ species volume fraction
angle of attack
angle of closure
$\epsilon$ interface thickness in level-set method
specific turbulent dissipation
$\gamma$ ratio of specific heats, $c_p/c_v$
$\phi$ signed distance function
$\mu$ molecular viscosity
$\rho$ density
$\sigma_c$ cavitation number based on cavity pressure, $(p_c-p_i)/q_\infty$
$\sigma_v$ cavitation number based on vapor pressure, $(p_v-p_i)/q_\infty$
$\sigma_{ST}$ surface tension

Subscripts
$b$ reference to outlet conditions
c reference to cavity properties
reference to air/hydrofoil chord length
$D$ reference to diameter
dyn conditions with a removed gravity field
$LOC$ reference to local properties
$L$ references liquid species
$max$ reference to a maximum
$min$ reference to a minimum
$N$ reference to cavitator
$G$ references gaseous species
$v$ references vapor
$\infty$ reference to free stream
Superscripts

- \(a\) advective term
- \(p\) pressure term
- \(s\) indicates a specific species
- \(R\) references to the right state
- \(L\) references to the left state
- \(+\) references to the right state
- \(-\) references to the left state
ACKNOWLEDGMENTS

The author is grateful to Penn State ARL for sponsorship through the E & F Fellowship and the Office of Naval Research through the HSSV program funded through Dr. Kam Ng. Furthermore, author is also grateful for the assistance that family and past colleagues have provided through the years. Specifically, my parents for their persistence in success in school, Dr. Duque, from Northern Arizona University, for initiating an interest in research and computational fluids, Phillippe G. and Eric J., from GE Wind, whose guidance assisted me to become a proficient engineer, colleagues in the Computational Mechanics Division at ARL, for all their help throughout this work, including Frank Z., Warren B., David B., Rick M., Eric P., Bill M., Norm F., Mario T., and Jack P, the members of my thesis committee, including Dr. Morris, Dr. Noack, and Dr. Hill, whose guidance has driven me to improve my thesis deficiencies and for all their useful and blunt critiques. Also, Dr. Maughmer, for all of his invaluable help guidance over the years as my M.S. advisor and as a Ph.D. committee member, together with and his role in helping me develop a strong interest in engineering research. Dr. Peltier, for all of his patience and his added direction throughout this research. Dr. Kunz, for his insights and guidance that led to many of the concepts carried out in this work. Dr. Lindau, for all of his guidance, patience, insights, input, and mentoring he has provided throughout this research effort. His efforts had enabled every success of this research. Finally, to my wife, Marta, for all of her patience, strength, support, and enjoyment she has given me through my efforts making it possible to make it through.
1.1 Motivation

The modeling approach for multiphase flows is an important aspect of marine-vehicle simulations that remains limited with respect to design and analysis. In hydrodynamic problems, multiphase flows occur at the free surface, on vehicle parts operating near the free surface, and in high-speed vehicle concepts based on supercavitation. These applications, amongst others, encompass a need for an improved modeling capability. The general assumption is that better analysis methods can improve the developed watercraft with less risk and investment.

Numerous modeling approaches exist for multiphase flows, but are often limited to a small range of the operational envelope. Complex configurations, off-design conditions, and viscous-flow regions can be difficult for many of the current techniques. Such deficiencies limit the insight gained, thus, a large part of the design and analysis has to be addressed in the experimental phase. In experiments, model adjustments for parameterization studies can be difficult and expensive; furthermore, proper model scaling is not always practical to achieve. The overall goal of this research is to improve the analysis at a stage prior to experimentation using a less limiting modeling approach, which is possible through incorporating additional physics in the modeling methodology.

Recent growth in computer technology and processing power has naturally led simulation advancements to consider more detailed fluid-flow simulations. These methods, sometimes referred to as computational fluid dynamics (CFD), use basic flow-
field assumptions that overcome some of the shortcomings that traditional modeling methods encounter. The caveat is that multiphase CFD methods are computationally expensive and not as developed as the single-phase methods, yet superimpose additional complexities. Furthermore, extending the simulations to handle complex geometries and configurations is also important; the modeling of such complexities is where CFD is most attractive. This extended capability is essential for improving the understanding prior to experiments.

1.2 An Introduction to Cavitation

The most general description of cavitation refers to gaseous cavities existing in a liquid flow. The gas is introduced artificially through various ventilation mechanisms or via phase change. Both are important for design and analysis, thus, they are introduced. Cavitation is common to many applications. In marine applications, it affects propeller performance, vehicle control, drag (reduction or penalty), noise levels and surface damage. Cavitation also arises in other engineering applications, such as in cryogenic-rocket propulsors, pump design, nuclear reactor design, etc. The practical application of cavitation extends to even the biomedical field, for example the shattering of kidney stones. Finally, cavitation exists in naturally occurring systems and processes. For instance, snapping shrimp utilize the collapse event of a cavitation bubble to stun or kill their prey [1]. This introduction focuses on common inception methods and physical aspects of cavitating flows in marine applications; it is intended to display a portion of the modeling requirements for an effective analysis method for vehicle design.
1.2.1 Vaporous Cavitation

Vaporous cavitation is a class of the boiling and condensation process. In normal boiling-condensation processes, the temperature varies about the saturation temperature with only small pressure changes. These temperature fluctuations are due to heating and cooling of the liquid. In vaporous cavitation, the liquid vaporizes and condenses by local pressure changes about the saturated vapor pressure and only small temperature changes occur. These pressure changes are a result of accelerating and decelerating a fluid. This most basic description of cavitation is sufficient for large-scale cavitation.

A limited number of parameters are important for large-scale cavitation. The vaporous cavitation number, defined as $\sigma_v = (p_\infty - p_v)/q_\infty$, relates the vapor pressure, $p_v$, to the free-stream dynamic pressure, $q_\infty = 0.5\rho V^2_\infty$, and the free-stream pressure, $p_\infty$. This cavitation number can be compared to the pressure coefficient, $c_p = -(p_\infty - p)/q_\infty$. One may expect cavitation to occur in flows where $-c_{p,\text{min}} > \sigma_v$. The Reynolds number, $Re = \rho VL/\mu$, is important in relating dynamic similarity as well as the resulting pressure field. Here, $L$ is a characteristic length of the problem. The Froude number, $Fr = V(Lg)^{-0.5}$, encapsulates the scaling of gravitational effects, or buoyancy. In addition, the Weber number, $We = \rho V^2 L/\sigma_{ST}$, is used as a scaling of the surface tension, $\sigma_{ST}$, which is negligible for many aspects of the large-scale cavities of interest. When considering cavitation inception, factors such as boundary-layer characteristics, water properties, and air content in the water all act as nucleation sites, thus, the criteria to incept cavitation. These nucleation sites are where phase change initiates; they may occur at dissolved bubbles,
turbulent eddies, or particles. The existence and amount of nucleation sites affects cavitation in general, but is not a significant factor for large-scale cavitation.

There are three classes of large-scale cavities, vortex, sheet, and cloud cavitation [2, 3]. Vortex cavitation occurs in the low-pressure regions of vortical flow structures. An example is displayed in the tip vortex in Fig. 1-1. Other vortical structures, such as those occurring in turbulent boundary layers, separated wakes, etc., also fall into this class. Sheet cavitation occurs at the low-pressure regions at the leading edge of a lifting surface or body. The cavity appears as a rather smooth sheet-like cavity; an example is shown just upstream of the tip vortex in Fig. 1-1. “Cloudy-looking” cavities that occur in a bubbly mixture are referred to as cloud cavitation. In Fig. 1-1, cloud cavitation is present at the aft, inboard portions of the wing. Although these guidelines distinguish the cavitation types, the defining characteristics tend to be inconsistent in that most large-scale-cavitating flows exhibit features that classify them into multiple types. Thus, these categories are merely a descriptive guide rather than a rigid categorization.
1.2.2 Artificial Cavitation

Artificial, or ventilated, cavitation differs from the vaporous form in that the cavity is composed of a noncondensable gas injected into the cavity to simulate the vaporous type. This is common for drag reduction concepts where the cavity encloses most of a body, thereby, virtually eliminating the skin friction drag. The most common configuration for marine vehicles consists of a cavitator, which is device at the vehicle fore body that uses a sharp aft-facing edge to purposely separate the flow. Air is ventilated into the wake of the cavitator, which eventually inflates into a large stable cavity that enshrouds the remaining part of the vehicle. Fig. 1-2 displays an example of an artificially created supercavity forming behind a cone-shaped cavitator [4,5,6].
A submerged supercavitating vehicle normally has a well-defined cavity-water interface that extends from the cavitator edge, to a location at, or past, the length of the vehicle. The cavity can terminate with a stable pattern or a turbulent-bubbly mixture. In either case, the cavity can exhibit a range of discrete and turbulent interfaces. This variability poses a challenge to modeling methods.

The nondimensional parameters in artificial cavitation differ from those of naturally cavitating flows. In this case the cavitation number, $\sigma_c = (p_\infty - p_c)/q_\infty$, is defined using the nearly constant cavity pressure rather than the vapor pressure. In general, the cavity size inversely relates to $\sigma_c$. In a time-averaged sense, for a steady-state cavity, the supplied gas rate into the cavity is equivalent to the rate of entrained gas from the cavity. It is typical to use the nondimensional ventilation rate defined as $C_Q = Q/(V_\infty D_n^2)$, where $Q$ is the ventilation rate, $D_n$ is the cavitator diameter, and $V_\infty$ is the free-stream velocity. The drag coefficient of a supercavitating vehicle is often calculated using a linearized drag behavior, from the drag value where $\sigma_c = 0$, or $C_D = C_{D,\sigma_c=0} + \sigma_c \frac{dC_D}{d\sigma_c}$, which behaves as

Figure 1-2: A glassy supercavity that forms behind a conical-shaped cavitator and enshrouds the silver-colored body. The photograph is borrowed from Stinebring et al. [5].
\[ C_D = C_{D,\sigma_c = 0} (1 + \sigma_c) \] for a disk-shaped cavitator. The \( C_Q - \sigma_c \) relation gives an understanding of the cavity size versus ventilation rate and reveals the efficiency of the ventilation system. A sample \( C_Q - \sigma_c \) plot is provided in Fig. 1-3. The \( C_Q - \sigma_c \) behavior is a strong function of the Froude number, \( Fr_D = \frac{V_\infty}{\sqrt{gD_n}} \), whereas similarity in the cavitator geometry is achieved using a drag equivalent to that of a disk cavitator. These curves define the cavity behavior using nondimensional parameters.

Figure 1-3: Typical ventilation rate relations. Solid lines represent the \( C_Q - \sigma_c \) relation at a given \( Fr_D \). Dashed lines represent the constant \( \sigma Fr \) values. Diagrams of each closure type are displayed to the right. The cavity-shape diagrams are borrowed from Semenenko [4].
The off-body cavity closure pattern can vary dramatically in supercavitating flows. This closure behavior is important as it dominates the air entrainment rate from a cavity. As illustrated in Fig. 1-3, three distinct cavity types exist and are referred to as twin-vortex, toroidal, and pulsating cavities. The twin vortex regime occurs when \( \sigma_c Fr < 1 \); this is the most studied closure type as a large and stable cavity is formed with the lowest \( \sigma_c \) values. Such cavity closure has two axial-aligned counter-rotating vortices that form at the aft end of the cavity. An example is shown in the diagram at the top-right portion of Fig. 1-3. These vortices are created from the buoyant loads on the belly of the cavity and are partially countered from downwash associated with a lifting cavitator.

The second closure type has toroidal-shaped vortices (\( \sigma_c Fr > 1 \)) and occurs at relatively low \( C_Q \) values for a given \( Fr \). This closure is created by radial-inward velocities that occur at the cavity terminus and tend to continuously shed off toroidal-shaped vortices full of the cavity air. This closure type is displayed in the right-centered diagram of Fig. 1-3. Vehicle operation with a toroidal-vortex cavity closure is not the typical choice; this is a result of the generally smaller, unsteady cavities. These cavities can also be less effective in reducing drag and can cause vehicle control problems.

The third cavity type is a pulsating cavity, which also develops at low \( \sigma_c \) values. As displayed in the right-lower diagram of Fig. 1-3, the gas leaks via pulsating subcavities. The parameter \( \beta \), defined as \( \sigma_v/\sigma_c \), is sometimes used to define the stability of this closure mode; however, \( \sigma_v \) seems to have been introduced for convenience rather than physical significance [3,4,7]. Although \( \sigma_v \) has little meaning in artificially cavitating flows, scaling laws based on \( \beta \) have shown merit [7]. These pulsating cavities have led
researchers to inconsistent conclusions as to their cause, which range from ventilation mechanisms to requiring free-surface interactions [7, 8]. Based on a linearized cavity theory, a more developed understanding of the instability is presented in Paryshev [9]. In summary, the solutions correlate the cavity pulsations to cavity-pressure oscillations reflecting along the length of the cavity. The major similarity parameter is associated with the nondimensional time lag, $\tau_0$, of such a cavity pulsations. This time lag is computed as, $\tau_0 = \sqrt{\frac{12}{\gamma (Eu/\sigma_c - 1)}}$, where the Euler number, $Eu$, is given as $2p_\infty (\rho_L V_\infty^2)^{-1}$, and $\gamma$ is the ratio of specific heats. When the mechanisms exist to excite this instability, the pressure fluctuations occur at this time scale.

When examining near-body cavities, additional air entrainment mechanisms become important. Influences such as the cavity closure location, surface roughness, vibrations, and jets can all affect the air-entrainment rate. These are either additive or subtractive mechanisms to the off-body closure types. The $C_Q-\sigma_c$ relationship is modified by the cavity closure angle onto a body, $\alpha$, or $C_Q \approx C_Q(\alpha, \sigma_c)$. This is the angle that forms between the cavity interface and the wall at the closure location [4, 10]. When $\alpha=0^\circ$, the cavity closes with the same angle as the local geometry and the streamlines smoothly transition from the cavity interface to the geometry surface. Such closure is stable and the ventilation rates can be reduced. At the other extreme, $\alpha=90^\circ$, the streamline is perpendicular to the wall and becomes a jet impingement-type problem. In a similar manner as in the toroidal-vortex closure, the impinging jet becomes reentrant, penetrates the cavity, and entrains additional air. Other effects, such as cavitation surface roughness, vibrations, and ventilation jets tend to excite a turbulent cavity interface, thus, increases
the air-entrainment rate through the cavity walls. Considering that drag benefits are maximized with a minimal cavity clearance, modeling these behaviors remains important.

1.2.3 Cavity-Free-Surface Interactions

In the context of surface-craft design, modeling vehicle and flow-field interactions with the free surface is crucial as it affects the maximum speed and dynamics of novel hull forms. There is a growing need to improve the analytical tools used for vehicle performance, stability, loads, and strength. Recent modeling advances in the marine hydrodynamics community have enabled detailed simulations of ship-wave interactions, ship dynamic-stability analysis (sea keeping), and wave-induced loading events. The focus here is towards natural and artificial cavitation, which is discussed in the context of free-surface applications.

There are numerous design applications in the regime where cavitating and free-surfaces flows coalesce. In one example, referred to as natural ventilation, cavities are composed of a noncondensable gas that is entrained into the cavity from above the free surface. This feature is common for lifting surfaces used on hydrofoil ships that remain near the ocean’s free surface. In this case, the wake pressure falls below the atmospheric pressure to provide the necessary pressure gradient to ventilate the cavity. Air can be supplied through the wake of a surface piercing strut \([11,12]\), following water-entry events \([13]\), suction peaks on surface piercing objects, or even through cavitating tip vortices that ascend to the free-surface \([11]\). Other free-surface interactions remain of interest. The loads of submerged lifting surfaces or vehicles operating near the free
surface can be altered significantly, especially when cavitating. The major issue here is that the vehicle stability characteristics are also modified [11].

1.2.4 Other Applications

Multiphase flows are apparent in many applications, where some have very similar issues as cavitation. Ice accretion on an aircraft-lifting surface, fusion chambers using liquid-wall protection schemes, nuclear-power-plant pipe breaks, water hammer, offshore structure analyses, are just a few of the applications that involve similar multiphase flows. More efficient and inclusive models can improve the understanding and design in such applications. Although these areas involve similar needs, they are not examined in this research as they are beyond the scope of the intended applications.

1.2.5 Physical Features of Cavitating Flows

In addition to the previous cavitation examples, the characteristics of cavitating flows have other physical aspects that are crucial to consider in the modeling. The first major point deals with the large density change occurring across the fluid interfaces. In vapor-water cavitation the liquid to vapor density ratio, $\rho_l/\rho_v$, varies from unity, near the critical point, to 40,000, at room temperature. Air-water interfaces have a more modest density ratio of around 1,000. These sharp density changes are a characteristic feature of the multiphase flows of interest. Furthermore, these interfaces may have a discrete
character, i.e. sheet cavities, or can be more of a bubbly mixture, such as apparent in cloud cavitation. Such features must be factored into the numerical scheme.

From a macroscopic viewpoint, the sound speed in a bubbly mixture decreases significantly. This is not obvious, considering that both the gas and liquid phases have rather high sound speeds, ranging from 300 m/s to 1500 m/s. In a high-density-ratio mixture, however, the bulk sound-propagation speed decreases dramatically. The sound behavior for a vaporizing water mixture [3] can be computed as

\[ \frac{1}{\rho a^2} = \frac{\alpha_L}{\rho_L a_L^2} + \frac{\alpha_G}{\rho_G a_G^2} - \frac{(1-\alpha_L)\rho c_{pL}T}{(\rho_L h_{LG})^2}, \]

where the mixture sound speed is given by \( a \), the volume fraction by \( \alpha \) and the subscripts \( G \) and \( L \) represent the gaseous and liquid properties, respectively. This sound speed can be derived by taking the control volume responses to pressure changes [2,3] or using the eigenvalues of the linearized, multiphase form of the Euler equations [14]. Note that the third term on the right-hand side occurs only with phase change. The decreases in the sound speeds for air-water and vapor-water mixtures without vaporization are displayed as a function of the gas-volume fraction in Fig. 1-4. Note that the sound speed in the mixture regions is much lower than both of the bounding single-phase values are. The sound speed is roughly 20m/s for the air-water mixture and 3 m/s for the vapor-water mixture (at 20°C). The cause of this sound-speed reduction is a result of the density changes with respect to pressure changes, or \( d\rho dp = a^2 \). In a bubbly mixture with a large density disparity between the gas and liquid, the bubbles give the mixture a very compressible feature. Any small change in the pressure causes a small density change to both the gas and liquid. More importantly, the change in density is not necessarily
uniform amongst the species, thus, the local volume fraction changes. This fact implies a large change in the mixture density, or physically appears as a sound-speed reduction.

Figure 1-4: Sound speed as a function of air-volume fraction in air/water, vapor/water mixtures and when a vaporization process occurs.

The third term in Eq. 1.1 occurs only with mass transfer and tends to further lower the sound speed. In this term, it is assumed that all of the available water vaporizes. For example, when vaporizing room temperature water the sound speed falls well below 1 m/s. Here, the large density changes occur through vaporization and condensation, thus, any small pressure perturbation about the saturation pressure creates extremely large density changes. Such behavior also causes significant sound speed reductions. In general, multiphase flows with large density and sound speed differences are likely to have regions where the sound speed varies significantly.
The described attributes of cavitating flows present several modeling requirements. First, the ability to retain a sharp interface is important, as it prevents artificial air entrainment that affects the predicted cavity behavior. Second, the modeling approach must retain the ability to model the physics of air entrainment from the cavity, which includes bubbly mixtures as well. For example, an artificially sharp interface may model the wrong behavior. Finally, in cavitating-fluid flows the isolated water is incompressible, the isolated gas is only moderately compressible, and the bubbly mixture regions are highly compressible. All of these features are important and affect the validity of the numerical schemes.

1.3 Background of Interfacial Fluid Flow Modeling

Many fluid-interface simulation techniques exist, which can vary widely in their applicability. The methods can be divided into three categories: (1) Navier-Stokes (N-S) based analyses, where some modeled form of the full N-S equations are solved. Popular techniques include solving the Reynolds-Averaged Navier Stokes (RANS), Detached Eddy Simulations (DES), Large Eddy Simulations (LES), and Direct Numerical Simulation (DNS) based methodologies. These are considered the best methods for examining flow-field details and massively separated flows. (2) Approximated Navier-Stokes methods include solutions to an approximated subset of the N-S equations, which can significantly reduce the solution time compared to N-S methods. An example is the use of potential flow assumptions that use the much faster boundary-element computational analysis method. This decreased solution effort also has a limited
predictive capability and requires case-specific models. (3) An \textit{empirical-based} method typically uses empirical corrections to simplified analytic solutions. Given their ability to analyze configuration changes efficiently, while maintaining reasonable solutions, they remain the most powerful design-level modeling approach. These modeling techniques are reviewed in the context of free surface, cavitation, and supercavitation analysis.

1.3.1 Review of Navier-Stokes Based Methods

N-S-based approaches attempt to solve the full three-dimensional N-S equations. Due to grid and time-step size constraints, the time averaged RANS form is often solved. This approach enables the efficient simulation of high-Reynolds-number flows by modeling the small-scale dynamics, rather than resolving them. As these methods can simulate features in the viscous regions, they enable a detailed investigation of the flow physics that can be useful in the design efforts to consider effects that inviscid methods cannot. Unfortunately, as a direct result of the immense computational effort, parametric design-level studies are not always fast enough to support an efficient design process. In addition, many challenges arise in the prediction of interfacial fluid flows with N-S-based methods; these challenges are a result of the large density gradients at interfaces that traditional methods cannot handle. Several common methods used in the naval analysis to overcome the interface-related issues are reviewed.
1.3.1.1 Surface Methods

Some numerical methods explicitly track interfaces and can be referred to as surface methods. Amongst these methods, two of the most common for ship hydrodynamics are described below.

1) Surface-Tracking Methods:

In a surface-tracking approach, the mesh boundary conforms to and tracks the interface. This method has been used to accurately simulate free-surface flows [15], ice-accretion problems for aircraft wings [16], and singular-bubble flows [17]. The benefits of this method are accuracy and simplicity, which is a result of retaining single-phase methodologies by modeling, rather than resolving, the interface. Complex features are possible, such as those occurring in the depicted bubble flow in Fig. 1-5, or even ice accretion [16]. For free surface or bubble flows, the interface is modeled at the coincident domain boundary using a kinematic boundary condition that enforces no penetration and maintains a constant, prescribed, pressure. For ice accretion, the interface motion is governed by a time-dependent rate of phase change to ice. In general, the method is limited to flows where interfaces do not fold over themselves, which is not well suited for breaking waves.

Limitations of the surface-tracking approach prevent it from being a suitable method for cavitation. An approach similar to that used for ice accretion could be used to model the phase change occurring in cavitation, however, as in breaking wave problems the interfaces tend to fold over and cause difficulties for the scheme. For this reason, surface tracking is only practical for simulating a limited range of interfacial-fluid flows.
2) **Level-Set Method:**

The level-set method is a surface-tracking technique that can be used to handle fluid interfaces. This method has proven itself for capturing discrete interfaces for free surfaces [18], breaking waves [19,20,21], direct simulation of bubble dynamics [22], amongst many other applications [23]. The numerical benefit of the level-set approach is the ability to capture a interface with a complex shape and without diffusion. The method is coupled to the governing fluid-flow equations using fluid properties.

**Signed-Distance Based Level-Set Formulation:**

The standard level-set method solves a transport equation for a signed-distance function from the interface, \( \phi(x,y,z,t) \). This particular function represents a distance to the nearest location of a fluid interface. Thus, at the level set, or isosurface, where \( \phi=0 \) yields
the location of the interface [22,23]. Solving a transport equation for \( \phi \) evolves the interface with the fluid particles, and is solved as

\[
\frac{\partial \phi}{\partial t} + \nabla \cdot (\vec{V} \phi) = 0 \rightarrow \frac{\partial \phi}{\partial t} + u_i \frac{\partial \phi}{\partial x_i} + \phi \left( \frac{\partial u_i}{\partial x_i} \right)^0 = 0. \tag{1.2}
\]

For incompressible flows \( \frac{\partial u_i \phi}{\partial x_i} = 0 \), thus, the traditional level-set transport equation is yielded, or \( \phi + \vec{V} \cdot \nabla \phi = 0 \). Although not always respected in practice, the method requires the resolution of all scales; this is the only case where the level set of a transport equation describes an interface. When true, a transport equation yields a constant value along a streakline-defined interface. Then, positive values of \( \phi \) can represent one fluid while the negative \( \phi \) values represent the other. This particular feature limits the baseline approach to two fluids. In the traditional algorithms, \( \phi \) is first initialized throughout the computational domain. The equation couples to the governing equations through fluid property relations. For example, the density could be defined using

\[
\rho(\phi) = \rho_g + (\rho_i - \rho_g)H(\phi), \tag{1.3}
\]

and a similar definition is used to describe the viscosity. \( H \) is a smoothed Heaviside function that has equivalence to both a characteristic and volume-fraction averaging. \( H \) is computed based on the local distance from the interface as

\[
H(\phi) = \begin{cases} 
0 & \text{If } \phi < -\varepsilon \\
\frac{1}{2} \left[ 1 + \frac{\phi}{\varepsilon} + \frac{1}{\pi} \sin \left( \frac{\pi \phi}{\varepsilon} \right) \right] & \text{If } |\phi| \leq \varepsilon \\
1 & \text{If } \phi > \varepsilon
\end{cases} \tag{1.4}
\]
where a finite thickness of the interface of $2\varepsilon$ is implied. The values of $\varepsilon$ are normally constant throughout the domain and common practice specifies it as $3/2$ the cell size. A finite thickness interface is used to improve the numerical stability [23]. Note that this feature is convenient for high-density ratio interfaces as the numerical schemes are often less stable [22, 24]. As the transport equation is only valid at the interface, integrating it in time only maintains a consistent signed-distance function on the interface. This introduces the need to reinitialize the signed-distance function so that some physical meaning of $\phi$ is maintained. Through this mechanism, the level-set method virtually eliminates both numerical and physical smearing. This reinitialization procedure commonly uses a computationally efficient iterative procedure [22]. The procedure is based on the behavior of $\phi$ away from the interface, that is $|\nabla \phi| = 1$, which is a differential form of satisfying Pythagoras’ theorem (see Appendix A.1). This suggests that $\phi$ can be reinitialized by solving an additional partial differential equation, a form of the Hamilton-Jacobi equation, given as

$$\frac{\partial \phi}{\partial \tau} = \text{sign}(\phi)(1-|\nabla \phi|).$$

1.5

Improved methods for solving Eq. 1.5 exist that attempt to maintain a constant $H$ and require fewer iterations [25]. It is reiterated that the effect of this reinitialization step eliminates any interfacial diffusion, both numerical and physical.

The benefit of the using signed-distance function as an interface-tracking variable is that it remains linear near the interface. This is not true everywhere, for example at points that are equidistant to multiple locations on the interface a discontinuous derivative occurs. This situation arises in the merging or breaking of an interface [27]. The use of
some type of upwind or shock-capturing method is necessary to prevent oscillations in those situations.

Level-set methods have been extended to compressible-fluid flows, but oscillations at the interface require complex algorithm additions. The inclusion of ghost cells near the interface are used to eliminate oscillations and to satisfy the correct jump conditions across an interface [26, 23]. Furthermore, mass transfer has been implemented into this method as discussed in the work of Sethian [23]. However, since the interface must be directly simulated tracked, all bubble scales in the mass-transfer process need to be resolved. This yields the method impractical for large-scale cavitating-fluid flows. Furthermore, the intermittent nature that can occur in cavitating problems creates difficulties in defining the signed distance function for situations where no vapor is present.

Alternate Variable Choices:

Recently, a conservative level-set approach was formulated in the work of Olsson et al. [28, 29]. Rather than a signed distance function, a regularized characteristic function (or volume fraction) is transported. Such a level set function, $\phi$, is bounded between 0 and 1 where the interface is located at the 0.5 level set. In this case, the fluid properties are based on a modified Heaviside function, where $H(\phi) = \phi$. Here the interface is again sharpened using a reinitialization step and is performed after each time step with the solution to

$$\frac{\partial \phi}{\partial \tau} + \nabla \cdot (\phi (1 - \phi) \hat{n}) = \varepsilon \nabla \cdot (\nabla \cdot (\phi \cdot \hat{n}) \hat{n}).$$

1.6
The term on the right-hand side is a dissipation term used for numerical stability, whereas the gradient term on the left-hand side is a compressive term that sharpens the interface in the interface-normal direction. The sharpening only occurs near the interface. This method is quite similar to solving volume-fraction transport equations (discussed below) with artificial compression terms \[30\]. Since this level-set formulation operates on conservative variables, it has the ability to conserve the volume of each species \[29\]. The fundamental difference of this formulation compared to the traditional signed-distance-function approach is through the sharpening approach. In the formulation using Eq. 1.6, the fluids are compressed in the interface-normal direction such that mass is conserved. Alternatively, in solving Eq. 1.5, the diffused parts of the species away from the interface are simply neglected.

**Overview of Method:**

Multiple issues remain with the level-set approach, especially in the context of cavitating flows. The most criticized issue relates to a lack of species-mass conservation. When the interfaces are highly deformed, such as those observed in a foaming-breaking wave, the basic assumptions are exceeded and mass loss tends to occur. Recent mass-conserving level-set methods appear to have addressed this issue \[29\].

Another concern arises in the application of the level-set method to cavitation problems pertaining to the need to resolve all of the bubble scales. For example, consider the time-accurate prediction of a breaking wave in Fig. 1-6 \[21\]. This is a level-set based LES simulation with highly resolved breaking-waves. In Fig. 1-6 (a), the initial wave is outlined by a single structure and is well predicted. At a later time, in Fig. 1-6 (b), the previously sharp interface breaks into many small-wave structures and bubbles. These
structures are nearly impossible to resolve computationally. As apparent in the experiments, three main waves are predicted using the level-set approach. However, the detailed structures of the actual wave-breaking event that is best represented as a foamy mixture, and is computed to behave in well-defined wave structures. Regarding applications, the impact loads imparted by a foamy wave strike compared to an artificially sharpened wave are expected to be quite different. This designed feature of maintaining a sharp interface presents a challenge and questions the application of the approach to many high-Reynolds-number interfacial-fluid flows. For example, cavitating flows with bubbly mixtures and foamy regions that affect air entrainment are important.

Finally, the accuracy of the momentum equations is of concern. The issue can be observed when taking the limit as $\varepsilon \to 0$, which yields $H$ values of zero or one and can no longer provide intermediate values. The resulting fluid property characteristic function, $H$, approaches a piecewise constant value over a cell implying first order numerics near the interface. Using sine-function smoothing yields higher order behavior without regard for physics. Special additions exist that can, in a physical way, account for the interface cutting through a cell to alleviate this modeling deficiency [31].
Figure 1-6: Level-set-based LES of a plunging-breaking wave compared to experiment. Simulations shown to the right with contour plots of vorticity and black lines indicate the predicted interface. (a) and (b) indicate different instances in time. All figures are borrowed from Yang and Stern [21].

### 1.3.1.2 Volume Methods

Volume approaches are based on the incompressible form of a species-mass conservation equation, resulting in the solution of a volume-fraction transport equation. The solution of such an equation, with a method of reconstructing and handling the
interface, provides a method suitable to handle interfacial-fluid flows. In the most general compressible form the species-mass conservation equation is given as

\[
\frac{\partial \rho \alpha}{\partial t} + \frac{\partial \rho \alpha u_i}{\partial x_i} = 0.
\]

Here, \( \alpha \) represents a volume fraction and \( \rho \) the density based on Amagat’s Law for a secondary species in the flow. For an incompressible flow, the density divides out and the equation becomes a transport equation of the volume fraction. In the context of a discretized domain, a cell with \( \alpha=0 \) may represent a cell completely full of water, and a cell with \( \alpha=1 \) would contain all air. Cells with intermediate values, \( 0<\alpha<1 \), indicate either the existence of an interface, a time-averaged representation of an interface, or a bubbly mixture of fluids. The volume fraction of each species is bounded by \( 0 \leq \alpha \leq 1 \), and only one transport equation is required for each additional phase. The remaining fraction explicitly determines the volume fraction of the other species; then the mixture densities can be used to satisfy a mixture-continuity equation. Note that such a formulation is equivalent to a mass-fraction-based method; however, for surface reconstruction a volume fraction approach is preferred.

A suitable definition of the interface using this variable is the \( \alpha=0.5 \) isosurface. This can be computed as a postprocessing step or using an embedded interface model. These multiphase flow formulations imply a locally homogenous mixture assumption, which intimates that the local velocity vectors of each phase are equivalent. This differs from the two-fluid approach, where each phase has its own velocity vector, pressure, and temperature. Such an approach results in each species requiring a new set of momentum and energy equations.
1) **Shock-Capturing Approach:**

The shock-capturing approach applies methods developed to capture shocks in gas dynamics to handle the large gradients occurring across a gas-water interface. An advantage of this approach is that the implementation is simple and exists in many CFD codes. The added requirement past a gas-dynamics code is the accounting for the additional species-mass transport equations with the corresponding mixture relations. Many approaches exist that are optimized for implementation into CFD codes. Furthermore, time-step size restrictions tend to be less limiting than those of other methods. Finally, the method remains suitable for broken and sharp interfaces and can handle a compressible fluid with no special additions. Compared to an explicit interface model, however, retaining a sharp interface tends to require more computational cells.

**Flux Limiting:**

Typical shock-capturing approaches rely on mixed upwind and higher-order differencing techniques. When done properly, this hybridized scheme prevents oscillations from occurring at discontinuous flow features. In these applications, a step-function-type behavior occurs in the density and volume fraction across an interface. A higher-order numerical approximation across an interface is not only oscillatory, but also, violates physical reasonableness implied by the Second Law of Thermodynamics. A simple, stable approach to alleviate this behavior is with a limited form of upwind differencing, which will enable a monotone, thus a Total-Variation-Diminishing (TVD) scheme. The total variation is an increasing measure when oscillations occur. Thus, a TVD scheme implies that the total variation is always decreasing. Furthermore, all monotonic interpolation schemes are TVD conformant. Limiting occurs using a variety of
techniques. Flux or slope limiters act by locally limiting the accuracy of the interpolation scheme near an interface; this is performed through a special averaging of the higher- and lower-order interpolation schemes. For example, a 1\textsuperscript{st}-order-accurate upwind TVD interpolation scheme is unacceptable due to low accuracy, whereas a 2\textsuperscript{nd}-order-accurate non-TVD linear interpolation scheme is inappropriate due to instabilities that develop across interfaces. The averaging provides a hybrid scheme that is both TVD and 2\textsuperscript{nd}-order accurate and is based on local derivatives used to detect discontinuities. Note that this is the only internal treatment of an interface within the algorithm.

**Inviscid Flux Formulations:**

Although upwind differencing approaches are effective, improved spatial differencing techniques can improve the prediction capability. At this point, it is useful to consider the unsteady, quasi-one-dimensional, Euler equations with an added species to describe how these methods are developed. The quasi-one-dimensional Euler equations, in vector form are

\[
\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} = H, \tag{1.8}
\]

where the conservative variables are included in the vector \(Q\), the flux vector, \(E\), and the source vector, \(H\), and are defined as

\[
Q = \left[ \rho A, \rho u A, e A, \bar{\rho} \alpha A \right]^T
\]

\[
E = \left[ \rho u A, (\rho u^2 + p) A, (e + p) u A, \bar{\rho} \alpha u A \right]^T
\]

\[
H = \left[ 0, p \frac{dA}{dx}, 0, 0 \right]^T.
\]
Here, the variables $\rho$, $u$, $e$, $p$, and $A$ correspond to the density, velocity, energy, pressure, and cross-sectional area, respectively. The second species density is given by $\tilde{\rho}$.

Returning to the numerical approach, the upwind direction must be determined. The traditional approach uses flux-vector- or flux-difference-splitting.

A flux-vector splitting algorithm splits the flux vector based upon the direction of the local eigenvalues, or the direction of disturbance propagation. For the system in Eq. 1.8, these correspond to the velocity, $u$, and the relative sound speeds, $u+a$ and $u-a$, where $a$ is the local sound speed. The flux vector is then split based on the direction of the eigenvalues, or $E = E^+ + E^-$, where $E^+$ contains influences from the positive eigenvalues and $E^-$ from the negative. Several references exist for uniform splitting procedures [32] and [33]. In general, after the flux-vector is split, and the solution equations become

$$\frac{\partial Q}{\partial t} + \frac{\partial E^+}{\partial x} + \frac{\partial E^-}{\partial x} = H.$$  \hspace{1cm} (1.10)

Now, the advection term is split based on the characteristic direction, where a stable, TVD scheme can be uniformly applied throughout the computational domain.

Advection Upstream Splitting Methods (AUSM) schemes are the most modern of the shock capturing approaches. A characteristic feature is that the flux vector is split into advection and pressure terms rather than using eigenvalues and the flux vector becomes

$$E = \begin{bmatrix} \rho u A \\ (\rho u^2 + p) A \\ h \rho u A \\ \tilde{\rho} \alpha u A \end{bmatrix} = E^a + E^p = u \Phi A + p \Psi A = u \begin{bmatrix} \rho \\ \rho u \\ h \rho \\ \tilde{\rho} \alpha \end{bmatrix} A + p \begin{bmatrix} 0 \\ 1 \\ 0 \\ 0 \end{bmatrix} A.$$ \hspace{1cm} (1.11)
Note that the flow work part of the energy equation remains in the advection term, thus, is expressed in terms of enthalpy. A discontinuity is assumed to occur at the cell face and the vectors $\Phi$ and $\Psi$ are expressed as functions of the left and right states as

$$E^{n}_{i+1/2} = U^+\Phi_L + U^-\Phi_R \quad \text{and} \quad E^{p}_{i+1/2} = P_{1/2}\Psi,$$

where $U^+$, $U^-$, and $P_{1/2}$, are determined using upwind-based polynomial functions of the algorithm-specific-defined Mach number. The formulations of the polynomials are described in Edwards et al. [34] for the Low Diffusion Flux Splitting Scheme (LDFSS), or in Liou [35] for the all-Mach number AUSM$^+$-up scheme.

Flux-Difference Splitting:

An alternate method is based on flux-difference splitting. This approach is sometimes considered more accurate and remains valid through highly compressible flows [32]. The benefits are at the cost of increased computational time and a less flexible algorithm. The discretized advection term is determined by differencing the flux across a computational cell. This is given as

$$\frac{\partial E}{\partial x} = \frac{\tilde{E}_{i+1/2} - \tilde{E}_{i-1/2}}{\Delta x},$$

where $\tilde{E}_{i+1/2}$ is the numerical flux on the right cell face and $\tilde{E}_{i-1/2}$ is the flux on the left face. For the time being, consider a first-order accurate model where the value of $Q(x)$ remains constant on the interval of the computational cell, $[x_i-\Delta x/2, x_i+\Delta x/2]$. Thus, at each cell face a discontinuity is implied. The numerical flux at the cell face, for the $i+1/2$ face, it is given as

$$\tilde{E}_{i+1/2} = \frac{1}{2}(E_i + E_{i+1}) - \frac{1}{2}(\partial E^+_{i+1/2} - \partial E^-_{i+1/2}).$$
The left terms imply a linear approximation and the right is the matrix dissipation. The $\delta$ terms in the matrix dissipation are defined by the difference in the flux vector, from $i+1$ to $i$. In an approximate Riemann solver, the linearization of the flux vector reduces the $\delta$-terms as

$$\delta E_{i+1/2} = E_{i+1} - E_i = \left. \frac{\partial E}{\partial Q} \right|_{i+1/2} (Q_{i+1} - Q_i) = \bar{A}_{i+1/2} (Q_{i+1} - Q_i), \quad 1.15$$

where the matrix $A$ is the flux-vector Jacobian defined as $A = \frac{\partial E}{\partial Q}$. The matrix $\bar{A}_{i+1/2}$ is the averaged Jacobian, which is a function of the solution-variable values at the cell face, thus, neighboring cells. This Jacobian must be appropriately defined in order to satisfy the relation in Eq. 1.15. A highly regarded method for obtaining $\bar{A}_{i+1/2}$ that exactly satisfies this relation was introduced by Roe [36]. This method works well for highly compressible single-phase flows, but the required constraints are difficult for multiphase flows with variable state equations.

Thus far, the description of flux-difference splitting only implies first-order accurate solutions. There are two common methods to achieve higher-order accuracy. The first is to represent the flow variables throughout the domain with a TVD-based, higher-order-accurate scheme. For example, the Monotone Upstream-centered Scheme for Conservation Laws (MUSCL), this TVD interpolation method yields cell-face variables from

$$Q(x) = Q_i + \varepsilon \frac{x - x_i}{\Delta x} \partial Q_i + \frac{3\varepsilon}{2\Delta x^2} \left[ (x - x_i)^2 - \frac{\Delta x^2}{12} \right] \delta^2 Q_i. \quad 1.16$$
In this equation, $\delta$ represents a linear variation in $Q$ and $\delta^2$ is the curvature of $Q$. The values of $k$ and $\epsilon$ control the order of accuracy of the scheme, which are modified internally to maintain a monotone TVD scheme. In terms of the flux-difference splitting scheme, as $Q$ is no longer assumed constant throughout the cell, the quantity $(Q_{i+1} - Q_i)$ becomes $(Q^R_{i+1/2} - Q^L_{i+1/2})$. Here $R$ indicates the cell right of the face ($i+1$ cell) and $L$ is the cell left of the face ($i$ cell). These face values are achieved using this MUSCL interpolation method.

Overview of Method:

In a density-based solution method, low-Mach number limitations exist. In the low-Mach number regions, the algorithm yields ill-conditioned matrices that require an infinitesimal time-step size. This deficiency introduces the need for preconditioning. For a Roe-based flux-differencing scheme, such preconditioning is required. Similar numerical schemes could be incorporated into a segregated-equation solution method, where such preconditioning is not necessarily needed. With a proper preconditioning approach, an all-Mach number formulation enables the simulation of the low- and high-Mach number flows apparent in cavitating flows.

In general, shock-capturing approaches are known to dissipate interfaces more than other approaches. Such dissipation artificially simulates mixing at the interface and suggests that high-resolution meshes are needed. Clearly, these added cells increase the computational effort. In cavitating flows, the cavity is normally confined to specific regions that are known ahead of the computation. Difficulties arise when trying to locally refining regions using a traditional structured-grid approach. Although several methods can be used for local refinement, approaches such as unstructured- and overset-grid
approaches are commonly applied. Local mesh refinement using shock-capturing methods has proven to be effective [37, 38]. Regardless, this added cell requirement is the major deficiency of the method.

2) *Volume of Fluid Method:*

The volume of fluid (VOF) method can be used to model a variety of large gradient-type problems. This method contains three distinct steps in the capturing approach [39, 40]: it approximates the interface, advects the interface, and internally applies boundary conditions at the interface. Excluding any of these is considered a pseudo-VOF method, which would include the shock-capturing approach [40] and perhaps even the volume-based level-set approach [29]. In the VOF methodology, the interface is modeled internally and is used to compute a numerical flux through each cell face. The impact of this internal model on the resolved interfaces is that they are captured in a single cell and do not diffuse artificially. This implies an improvement over the three cells required for the level-set approach, and three-five cells for a shock-capturing approach. The caveats of the VOF approach are that logical complexities introduced through the algorithm, low-Courant number constraints exist, and it is normally only suited for time-accurate simulations. Nevertheless, the method has been highly successful in simulating interfacial fluid flows.

**Standard Formulation:**

Numerous interface-reconstruction techniques exist. In general, the reconstruction of the interface is approximated using the local and neighboring volume-fraction values [41]. The original first-order reconstruction method uses a simple line interface calculation (SLIC), which is a grid-aligned approximation shown in Fig. 1-7 (a). Higher-
order methods apply a better approximation of the interface, which is displayed in the piecewise linear interface calculation (PLIC) shown in Fig. 1-7 (b). These higher-order reconstruction schemes can be useful, but also require iterative procedures that may be computationally expensive.

![Sample cell volume-fraction distribution with various interface reconstruction methods.](image)

**Figure 1-7:** Sample cell volume-fraction distribution with various interface reconstruction methods. The line represents true interface, shaded regions are the reconstructed interface computed with (a) SLIC and (b) PLIC schemes. Diagrams borrowed from Scardovelli and Zaleski [41].

Following this reconstruction, the interface is advected based on the locally computed flow field. Various methods are used to integrate the interface motion. Recent approaches, such as described in Scardovelli and Zaleski [41], use a Lagrangian-based interface motion algorithm. An example is shown in Fig. 1-8. The defined interface advects with the flow particles and the numerical flux of each fluid is computed based on the interface representation and motion from one cell to another. Complexities arise when considering multidimensional and non-Cartesian meshes. Some complexities can be avoided using a split-operator procedure, where an advection and reconstruction step occurs for each coordinate direction, at each time step.
Finally, boundary conditions are imposed at the interface. Such conditions allow the fluids to have different speeds that remain tangent to the interface. This essentially removes the homogenous-mixture assumption for velocity. The pressure can also be specified at the interface by interpolation from the reconstructed interface to the cell-value storage location. For free-surface problems, the pressure on the interface is a known value. However, in cavitating flows it is a result of the solution and is not typically prescribed.

**Overview of Method:**

The method is, in general, fully conservative and remains valid for large deformations and breaking waves. There are several drawbacks to this method, including the artificial breaking of the interface and time-step size constraints. As noted earlier, the surface is reconstructed based on the neighboring cells and imposes a limiting Courant number that is less than unity. This restriction is further implied with the use of the Lagrangian advection of the interface. Note that multiple integration cycles, or
subintegration steps, with a segregated solution procedure could increase the overall Courant number. With these Courant number restrictions, the time integration to steady state can increase.

On a final note, the method behaves well when considering high-density ratio interfacial-fluid flows. This is achieved through the interface advection step and the application of boundary conditions at the interface. In these two steps, the momentum equation is treated outside of the discretized governing equations. This eliminates the large momentum gradients at the interface and removes the destabilizing regions from the discretized momentum equations.

1.3.1.3 Combined Methods

Efforts to improve the weaknesses of the aforementioned methods have focused on combining different modeling techniques. The current algorithms use a Combined Level Set and VOF (CLSVOF) approach. This sophisticated method combines the strengths of the VOF and level-set approaches, which enables a mass-conserving method that uses the high-accuracy level-set method to reconstruct the interface [19,42]. In this method, two transport equations are solved for each added phase, as well as another PDE for the reinitialization of the level-set function. In the case of representing a fluid mixture, one must question the need to solve two sourceless transport equations that yield only a single variable, the volume fraction. It is argued that such a solution method is over specified and the computational time spent solving an additional equation does not yield additional information.
1.3.1.4 Grid Methodologies

Improved grid technologies are essential for accurately simulating interfacial-fluid flows. These improve the solution quality by local error reduction, reduce the effort in building the computational model, and enable complex simulation capabilities that include fluid-body or fluid-structure interaction.

1) Local Grid Adaptation:

Solution truncation errors depend on the local size of the computational cells and the magnitude of the higher-order derivatives. Grid adaptation is a method that automatically optimizes the mesh size based on local flow features; this includes local cell-size refinement and coarsening depending on some measure of error. In terms of interfacial fluid flows, any of the numerical methods discussed can be enhanced with Adaptive-Mesh Refinement (AMR). This is especially true near breaking waves where many flow scales exist. Previous applications of AMR were shown to enhance the shock capturing [37], VOF [43], and CLSVOF [19] approaches for free surfaces with breaking waves.

Although many AMR methods exist, an efficient octree-based method applicable to unstructured grids is reviewed as a general example. For simplicity, the two-dimensional equivalent quadtree method is displayed in Fig. 1-9 below. In such a refinement technique, the successive dividing of an original coarse grid is used for efficient mesh refinement. The original grid system is represented by the highest generation, or parent cell. The next generation forms four children as equal partitions of the parent. In cases where refinement occurs within the children, another generation of
four cells forms. An octree method differs in that eight cells are generated due to the third dimension. With appropriate solver logic, successive refinement of this sort can occur with specified mesh resolution based on a measure of error. Another requirement is that the solver must be capable of handling cells with an arbitrary number of cell faces. This is a result of the “hanging nodes” created in the process, which is displayed in Generation 2 in Fig. 1-9.

Figure 1-9: Description of a quadtree refinement strategy.

In the context of free-surface predictions, the AMR approach has been shown to yield highly accurate free-surface wave predictions [37,38]. An example of AMR applied to a breaking wave is displayed in Fig. 1-10. Note that the adaptation is concentrated only in the proximity of the free surface, which effectively reduces the artificial dissipation near the interface [38]. As the waves move, cells are created and destroyed such that the high-resolution mesh conforms to the dynamic free surface.
These adaptive methods are attractive in that they greatly reduce the ambiguity in the grid generation process and provide a method that can be adapted to resolve moving interfaces as well as turbulent structures. Furthermore, the accuracy of any numerical interface modeling strategy can be augmented by efficiently refining the cell resolution onto a resolved interface. The drawbacks of the adaptive method result from the added complexity in the solver logic and the difficulty associated with unbalanced mesh decomposition. Furthermore, there has been difficulty in achieving optimal refinement strategies \[37,43\]. In theory, a coarse grid with refinement on the interface should provide a similar quality solution as a well refined grid throughout the domain. As observed in both the work of Hay and Visonneau \[37\] and Rhee \textit{et al.} \[43\], this is not easily accomplished by adaptation to the interface, suggesting a lack of understanding in the effective use of such methods.

![Figure 1-10: Example of adaptive-mesh refinement applied to a breaking wave. Figure borrowed from Hay \textit{et al.} \[38\].](image)

2) 

\textit{Overset-Grid Approach:}
The overset-, or Chimera-, grid approach [44] is a powerful mesh technique that enables solutions on a computational domain that does not have face-matching requirements. The idea is to cut out “holes”, or remove cells, that exist within solid bodies. The boundaries of these “holes,” as well as the edges of other domains that may exist in the composite grid system, use interpolations that recouple the systems of unmatched, overlapping domains. The strength of the method is to accommodate moving-body applications as well as relieve point and face matching constraints. The main disadvantage is that mass conservation is no longer guaranteed.

In the context of interfacial-fluid flows, overset has been used extensively for ship-wave interactions [18]. A refined overset mesh can be strategically positioned to resolve features of interest, including bow waves, wakes, etc. This can be used for refinement throughout the domain and assist to improve the resolution of interface structures of interest. Another example of overset grids applied to interface modeling, as displayed in the simulated Hele-Shaw flow (see Fig. 1-5 above). Here, a surface-tracking mesh handles the interface. The overset mesh enables a method that can adapt with the interface and enables complex motion of the bubble within the medium. This relates to the specific interest of improved simulation capability, where the overset-grid approach enables dynamic motion without the complexities required in a deforming-mesh approach [18]. These are the main advantages of the overset-grid method.

3) Cartesian Methods:

Cartesian grid methods are an attractive approach if the solver can locally adapt to flow features, such as an interface and solid bodies. These methods tend to be more robust, accurate, and require less computational time per a cell. As the grids remain
Cartesian, body-fitted grids are only practical for a limited number of geometries. Thus, some sort of representation of the bodies is required and is done using embedded boundary conditions. Resolving all of the significant physics, i.e. high-Re number boundary layers, remains challenging for this technique without such adaptation [45].

1.3.1.5 Turbulence Modeling

A RANS approach is achieved using ensemble averaging of the N-S equations. The benefit of this ensemble averaging is the isolation of the high frequency turbulent fluctuations from the mean values. This introduces the need to close the governing equations with turbulence models of these turbulent fluctuations. The turbulence models are extremely important for CFD methods in that they enable reasonable predictions of high Reynolds number flows. Here, a model for the mean turbulent fluctuations, or Reynolds stresses, is used to close the system of equations. Based on the Boussinesq-eddy-viscosity assumption, an effective eddy viscosity due to the turbulent fluctuations is used to encapsulate the effects of turbulence. The current standard bases the eddy viscosity on one- or two-equation models of the transport, production, and destruction of various turbulence parameters. Typically, in some analytic fashion, these parameters relate directly to the turbulent kinetic energy and turbulence dissipation rate [46]. For the interfacial fluid flow applications of interest, it is quite possible that predictions depend on the modeling of the turbulence behavior.
1.3.2 Review of Approximated Navier-Stokes Methods

Many of these methods have been the traditional approaches to handle interfacial flows, and are thus well developed. They are based on solutions to an approximated form of the Navier-Stokes equations that greatly reduce the required computational effort. These methods are extremely powerful in the design context, and must be considered as a tool in order to arrive to an optimal solution strategy.

1) Reduction of Dimensions:

The shallow-water wave (SWW) equations are an approximate form of the Navier-Stokes equations, where a reduction in the gravity-aligned dimension is used to model free-surface type flows. Among other solution techniques, these equations are solved directly using shock-capturing-based CFD methods. In its derivation, the incompressible, inviscid flow assumptions are invoked. Using a column-shaped differential element that span from the floor to the free-surface elevation, \( h \), waves having a wave number, \( k \), can be modeled. Here, all velocities in the \( z \) direction are neglected. This assumption is completely valid for shallow-water waves, waves where \( kh << l \), as the waves induce elliptical motions beneath the free surface. This is in contrast to the circular motions created in deep-water waves [47].

The 2D+t method is based on slender-body theory developed for load prediction on aircraft fuselages and low-aspect ratio wings. It relies on the assumption that axial-direction derivatives are small. Based on this, one can treat the axial direction in time using \( x = Ut \), rather than another spatial dimension. The method has been applied to slender ship hydrodynamic analyses, in particular modeling bow waves, for both
boundary-element and CFD methods \[48,45\]. For a CFD-based analysis, this significantly reduces the overall computational time by minimizing the number of cells required in a simulation. The method is only valid for thin bodies at high Froude numbers. A sample prediction using this method is displayed in Fig. 1-11 below.

![Figure 1-11: Computations of free-surface predictions using the 2D+t methodology. Each slice is a prediction at another instance in time, which is modeled to represent the stream-wise direction. Figure borrowed from Andrillon and Alessandrini [48].](image)

2) Physical Approximations:

The potential-flow assumption is characterized by an incompressible, irrotational, inviscid flow. Boundary element, or panel, methods are often used to solve this reduced set of the N-S equations. In the method, only the boundaries are discretized rather than the entire flow field; this enables a more efficient method that maintains a high-accuracy, low-dissipation numerical technique. These methods are also efficient for design-level modeling while retaining the ability to yield inviscid flow field insight.

Potential-flow methods have been used as a method to analyze many types of cavitation and free-surface flow problems that include artificial supercavitation \[4\], large-
scale cavitation [49], surface piercing hydrofoils [49], and free-surface problems [49, 47]. In cavitation problems, the potential flow assumptions are reasonable in all regions except within the cavity, boundary layer regions on the cavitator, and in the cavity closure regions. Such features must be modeled. In free-surface type problems, with the exception of breaking wave regions, the flows also behave in accordance with potential flow assumptions.

All of the deficits of the potential flow modeling approach are due to the need to model the viscous portions of the flow. Shear layers handled with boundary layer models are well developed and are very accurate for attached and mildly separated flows. Such models are more difficult to develop for a broad range of flows, especially for large-scale separation. For cavitating flow predictions, a constant cavity pressure is generally assumed; this is a valid assumption that removes the turbulent cavity air from the prediction. The cavity closure heavily relies on closure-specific models that cause the predictions to rely heavily on code calibration. Similar situations arise for free-surface problems, where smooth waves are predicted well, but breaking waves become difficult to model.

1.3.3 Empirical Methods

Due to their simplicity, empirical methods have been the preferred approach to model problems in the design context. Such methods often include the combined ability to make configuration changes based on altering a single input parameter and predicting some feature in a fraction of the time of more complex analysis methods. Empirical
approaches also efficiently integrate into vehicle dynamic analyses. These methods reduce the computational overhead to predict physical behavior, while maintain a reasonable amount of accuracy for initial design. Such an example is given for artificial supercavitation.

As previously discussed, in steady supercavitation the air is supplied at the same rate it is leaked. An established empirical model that predicts the rate of air entrainment for a cavity with twin-vortex closure is given as

\[
C_g(\sigma) = \frac{0.45 c_{D,0}}{\sigma (\sigma^3 F r^4 - 2.5 c_{D,0})}.
\]

In a steady ventilation conditions, this relationship yields \( \sigma \) for a specified air ventilation rate. In addition, an analytic solution to the potential flow formulation can be determined when assuming an elliptical cavity shape and that \( \sigma_c = 0 \). Using this analytic solution as a baseline, with only a few empirical modifications it has been generalized to predict the cavity shape for all twin-vortex cavities [4]. The cavity shape directly relates to cavitator parameters such as the drag, \( c_D \), radius, \( R_n \), and the cavitation number. The cavity shape parameters include the cavity radius, \( R_c \), length, \( L_c \), and the axial profile, \( R(x) \). Thus, many parameters of interest can be explicitly calculated from only a few integral parameters. This clearly enables efficient design-level computations. For instance, using a momentum-theorem based analysis [50], and several empirical parameters, the cavity shape can be predicted to behaves as
where $k$ and $A$ are empirical parameters that are roughly 1.0 and 2.0, respectively. Note that the disk-cavitator drag coefficient behaves roughly as $c_D = c_{D,0}(1 + \sigma)$, with $c_{D,0}$ being the drag when $\sigma = 0$, and using a value of $x_i = 2R_N$ is typically. Finally, the axis deflection resulting from the combined effects of the cavity buoyancy and downwash is calculated using [4]

$$h(x) = \left[ \frac{(1 + \sigma)x^2}{3F_{r_i}^2} \right]_{\text{gravity}} + \left[ -c_L R_n (0.46 - \sigma + x/R_c) \right]_{\text{Lift}}.$$  \hspace{1cm} 1.19

The separate gravity and lift terms are indicated with subscripts; note that $F_{r_i} = V_{\infty}/\sqrt{gL_c}$ and $c_L$ is the cavitator lift coefficient. Other approaches exist that extend such methods to predict unsteady cavities [4].

Within their many limitations, these relations can efficiently supply the cavity shape and dynamics. From a design standpoint, this is useful in that geometric adjustments are trivial, the computation time is low, and the accurate predictions are still achieved. Unfortunately, the approach lacks the ability to model important detailed flow features. In addition, limitations arise with complexities in cavity-body interactions, control surfaces, and propulsors. These details remain a crucial aspect of supercavitating vehicle design that can only be examined experimentally or with a more detailed analysis method.
1.3.4 Hybrid Methods

Hybrid solution methods combine two or more of the previously discussed approaches. Such an example was presented by Iafrati and Campana [51], labeled as a domain decomposition method, which was used to handle breaking waves. Here, the domain is decomposed in a way that takes advantage of the strengths of each method. Consider the problem in Fig. 1-12, where the waves are generated in a channel flow over a bump. A panel method is used to model the flow field well below the surface, indicated by $\Omega^B$. Here, potential flow assumptions and models are sufficient as long as the assumptions are not exceeded. In the proximity of the free surface, region $\Omega^F$, a CFD-based approach using the level set method is used to handle breaking waves. In this case, the flow is better handled by a CFD approach. The two solution methods are coupled at the surface $\Gamma$ [51]. Although the method is limited; it is efficient and effective for a range of flows, including streamlined and submerged vehicles or objects.

![Diagram](image)

**Figure 1-12:** Domain decomposition method, a level-set based CFD approach is used in the region $\Omega^F$, and a potential flow is used in $\Omega^B$. Coupling of the domains is made at $\Gamma$. Diagram borrowed from Iafrati and Campana [51].
1.3.5 Summary of Reviewed Computational Methods

An overview of the reviewed analysis methods is given in Table 1-1. It also details a brief statement on the benefits and case-specific challenges for each method. It is clear that no current approach provides a solution method applicable across all the naval applications of interest. Arriving at the ideal approach will likely include a variety of the approaches in existence.

Table 1-1: Overview of the approaches to handle interfaces.

<table>
<thead>
<tr>
<th></th>
<th>General Pros</th>
<th>Ship Hydrodynamics Cons</th>
<th>Cavitation Cons</th>
<th>Supercavitation Cons</th>
<th>General Cons</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface Adapting</td>
<td>Only retains sharp interface</td>
<td>Cannot handle breaking waves</td>
<td>Requires closure modeling, no benefit over boundary element</td>
<td>Requires closure modeling, no benefit over boundary element</td>
<td>Limited to interfaces that do not fold over</td>
</tr>
<tr>
<td>Level Set</td>
<td>Maintains sharp interface, no geometric constraints</td>
<td>Must resolve all scales in breaking waves</td>
<td>Must resolve all scales in bubbly mixtures</td>
<td>Must resolve interface mixing, or model air entrainment</td>
<td>Not conservative</td>
</tr>
<tr>
<td>Shock Capturing</td>
<td>Maintains interface, fast time integration, extends to compressible flows</td>
<td>Dissipative free surface</td>
<td>High resolution required</td>
<td>Dissipative surface</td>
<td>Dissipative interface, surface tension difficult to add into model</td>
</tr>
<tr>
<td>VOF</td>
<td>Maintains sharp interface, no geometric constraints, conservative</td>
<td>Often run time accurate, CFL limitations</td>
<td>Cloud cavitation must rely on shock capturing</td>
<td>Closure must rely on shock capturing</td>
<td>Surface tension inaccurate</td>
</tr>
<tr>
<td>CLSVOF</td>
<td>Maintains sharp interface, no geometric constraint, conservative</td>
<td>Often run time accurate</td>
<td>Cloud cavitation must rely on shock capturing</td>
<td>Closure must rely on shock capturing</td>
<td>Complex algorithm</td>
</tr>
<tr>
<td>Potential Flow</td>
<td>Quick solution time, retains sharp interface, nondiffuse</td>
<td>Cannot handle breaking waves</td>
<td>Cannot handle cloud cavitation</td>
<td>Relies on models for closure, complex geometries</td>
<td>No Evaluation</td>
</tr>
<tr>
<td>2D+T</td>
<td>Reduced solution time, enables high resolution</td>
<td>Can not handle low Fr, wide-bodies, or traverse waves</td>
<td>N/A</td>
<td>Limited to specific closure modes, requires modeling</td>
<td>No Evaluation</td>
</tr>
<tr>
<td>SWW</td>
<td>Reduced solution time, always retains sharp interface</td>
<td>Not well suited</td>
<td>N/A</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>Empirical Based</td>
<td>Extremely fast, reliable, easily configurations changes</td>
<td>N/A</td>
<td>No Evaluations</td>
<td>Reveals few details, relies on empiricism</td>
<td>N/A</td>
</tr>
<tr>
<td>Hybrid Methods</td>
<td>Computationally efficient with little or no loss of accuracy</td>
<td>N/A</td>
<td>Difficult to apply in wake of cavity</td>
<td>Difficult to apply in wake of cavity</td>
<td>Complex algorithm</td>
</tr>
</tbody>
</table>

Considering the applications of interest, only the volume-based methods appear appropriate for supercavitation problems. This is a result of the requirement of the mixed sharp/broken interface modeling requirements. A level-set approach incorporating a
cavity-interface air-entrainment model could be considered for cavitation, but it requires an additional modeling step. Finally, in comparison to a VOF and shock-capturing technique, it is argued that a shock-capturing technique is more flexible in almost every respect in the modeling algorithm, which includes mass transfer modeling, extensions to n-species formulations, compressible formulations, Courant limitations, local time stepping, and error propagation from domain. In comparison, the VOF approach has lesser grid requirements, roughly three times as few in the regions near an interface. In the context of an adaptive grid algorithm, interface refinement with shock capturing would be an adequate extension. There are, perhaps, avenues to improve the overall modeling capabilities through modifications of current methods.

1.4 Thesis Scope and Outline

1.4.1 Scope of Thesis

Traditional CFD approaches cannot handle the many complexities that occur in real-world multiphase simulations. Difficulties tend to arise when simulating intercomponent interfaces, most notably numerical instabilities and nonphysical diffusion of the interface. These issues have been overcome within the CFD community with model development and numerical techniques. Specific advancements include improved models of interfaces and turbulence, as well as enhanced numerical schemes with preconditioning that enable simulations with more accurate physics. Real-world geometric configurations can be difficult using a traditional computational mesh
approach. These difficulties arise at geometric complexities, moving parts, or when examining vehicle dynamics. Many modern mesh approaches exist that enable the simulation of the aforementioned complexities. These existing methods are expected to enable the simulation capability needed for supercavitation design and analysis.

In these studies, the goal is to survey and promote modern computational techniques that enable an adequate simulation capability for cavitation without limiting the ability to examine free-surface interactions. Attention is focused on modeling fluid interfaces, which is a salient feature in the flows of interest. In these studies, a broad range of the CFD algorithm is examined in hope to improve the numerical prediction capability. The studies range from investigating the numerics to the modeling approaches. As it is difficult to examine all available methods, the scope will be limited to examining the following in detail:

1) *Numerical Techniques:* This includes the evaluation and validation of the numerics, and improved computational methods that enable a better simulation and modeling ability. Such improved approaches are both based on mesh and numerical alterations.

2) *Physical Modeling Requirements:* Investigations of various physical parameters that may affect the predictions is performed, and are limited to density, turbulent, and compressible effects.

3) *Physical Discovery:* Using validated models, detailed features of the flow can be investigated, which are used to examine features that are difficult to understand using experimental approaches.
1.4.2 Outline of Thesis

Chapter 2 presents the governing and model equations, provides an overview of the code used for these investigations, and introduces the capabilities introduced into the code within this work. The governing equations for multiphase flows are first introduced and examined with respect to the validity of the homogenous-mixture assumption for cavitating and free-surface flows. The model equations, which differ slightly from the governing equations, include turbulence and cavitation modeling, as well as preconditioning. These are discussed along with the solution approach to these model equations.

Chapter 3 includes descriptions of the test cases used in the validation. These descriptions include the experiment setup and conditions. The chapter also includes discussion of the assumptions and meshes used in the simulations. As this chapter is referenced significantly throughout the remaining work, an understanding of these test cases is important.

In Chapter 4, the numerical methods are assessed on several test cases. Included in this assessment are evaluations of various numerical schemes, grid and time studies, and the validation of the numerical solutions using theory and experiments. These are all performed on relevant examples. The results from these studies give an excellent indication that the numerical approach is effective in the simulation of cavitating and free-surface flows.

Within the scope of this research, several additions to the numerical approach are presented in Chapter 5. An improved simulation capability enabled through overset grids
is displayed. Within the context of a structured-grid solver, an adaptive-mesh method based on a combined surface-tracking-VOF approach is presented. A key feature is that the adaptive method is conducive to the requirements for high-fidelity free-surface simulations. Finally, an improved numerical method for volume-based multiphase approaches is introduced, which is based on the level-set approach. These added capabilities are presented, tested on real-world cases, and show promise in the advancement towards better simulation techniques.

In Chapter 6, the physical modeling requirements for the examined multiphase flows are explored. The studies include common physically based assumptions and their regions of validity. Included in these efforts are various approaches of modeling turbulent and viscous flow. These studies vary from wall-bounded boundary layers, interfacial boundary layers, and separated regions of the flow field. Finally, modeling sound-speed reductions within the cavities and phase-change-induced temperature effects are also scrutinized. These studies display situations where the assumptions and modeled physics affect the predictions.

Finally, in Chapter 7, the CFD predictions are used to gain additional insight into the physical behavior of cavitation. The present investigations verify theories of the mechanisms of air entrainment based on boundary-layer behavior within the cavities. Furthermore, investigations of hysteretic cavities are examined and display future model requirements for better simulations in these regimes. Cavitation-free-surface interactions are examined and validated, which show to influence the free-surface signature of a vehicle. Finally, cavitating hydrofoils with oscillations are investigated.
1.4.3 Contributions

Throughout this research, various approaches are explored to improve the CFD analysis of multiphase flows. This includes advancements in both predictive and simulation capabilities for supercavitating-fluid flows. First, the baseline algorithm is validated and an improved understanding of the necessary numerics is developed. Second, capability-extending meshing algorithms are evaluated for solution improvement and adding dynamic simulation capabilities. The methods used are based on the overset grid approach and adapting meshes. Next, a more effective numerical scheme based on the level-set approach is developed and validated. Contrary to previous formulations, the method consistently extends from the baseline form to handle compressible flows, flows with phase change, and an arbitrary number of species. Additional reinitialization techniques are also derived that are trivial to implement and are more representative of the interfaces occurring in cavitating flows. The effects of various physical phenomena are also examined based on the computational predictions. These effects are shown to be crucial in the accurate prediction of specific regimes of a cavitating flow. In general, the overall results demonstrate an accurate simulation capability for cavitating and free-surface flows, with the additional abilities to simulate the many complexities desired in real-world problems. Finally, added insight into cavitating flows is extracted with the use of detailed flow-features that CFD solutions provide.
Chapter 2
Theoretical Model Formulation

The physical, modeled, and computational-solved equations are discussed in this chapter. First, the governing equations are introduced and approximations are examined to arrive to the homogenous-mixture model. Next, other models are presented that describe the turbulence and phase change behaviors. Following this modeled set of equations, is the form used to solve numerically within the CFD code. Later, are summaries of the contributed capabilities to the CFD code within the scope of this work.

2.1 Governing Equations

This section describes the governing equations of multiphase-fluid flows and the modeling approaches used for subgrid scale behaviors and cavitation. In the most basic form, the governing multiphase equations are identical to those for single-phase flows with state equations valid across all species. Using such a complex state equation can be difficult to develop and computationally expensive in algorithms that include numerous state-equation references. Alternatively, as presented here, conserving the mass of each species with different properties and state equations is just as valid for multiphase flows. A review of such a modeling approach is considered, from the continuum model equations to a form that includes the subscale multiphase interactions. This is followed by an analysis of the regimes that become important and leads to the homogenous model.
2.1.1 Multiphase Flow Modeling

The differential governing equations for multiphase flows parallels those for single-phase flows. Additional species are naturally accommodated through species-mass conservation equations. The conservation of mass for \( n \) species, with mass transfer, are given by

\[
\frac{\partial (\tilde{\rho}\alpha_s)}{\partial t} + \frac{\partial (\tilde{\rho}\alpha_s \, u_i)}{\partial x_i} = \dot{\omega}^s, \text{ for } s=1 \text{ to } n, \tag{2.1}
\]

where the superscript, \( s \), denotes the species-dependent value for the volume fraction, \( \alpha \), density, \( \tilde{\rho} \), and the rate of mass transfer, \( \dot{\omega} \). The individual species densities are specifically based on the local pressure and temperature, as appropriate for Amagat’s Law, rather than the partial pressure as used in Dalton’s Law. Note that the mass-transfer rate for one species indicates that another source, with an opposite sign, exists in another species. The mass transfer is omitted for this discussion, i.e. \( \dot{\omega} = 0 \).

In the continuum, the momentum equations are described using the properties of the local species that result in the standard Navier-Stokes equations, when surface tension is neglected, and are given as

\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right). \tag{2.2}
\]

Here, the local fluid-dependent properties for the density, pressure, viscosity, and velocity are given by \( \rho \), \( p \), \( \mu \), and \( u_i \), respectively. Note that this set of momentum equations is identical to that for single-phase flows. The energy equation is formulated in a similar sense, again based on local, fluid-dependent properties. In total, the
conservation of mass yields \( n \) equations, while the mixture momentum and energy equations introduce 4 more.

Finite resolution limits introduce the need for additional modeling. In the most general form, phase-specific flow variables (i.e. \( p, T, u_i \)) are assumed resulting in a series of \( n \) systems of five equations. Again, each of these systems closely resembles those for single-phase flows. The species conservation equations remain consistent with Eq. 2.1. As presented by Kunz et al. [52], the momentum equations for a multifluid system for bubbles being carried by water are given by

\[
\frac{\partial (\tilde{\rho} \alpha_i u_i)}{\partial t} + \frac{\partial (\tilde{\rho} \alpha_i u_i u_i)}{\partial x_j} = - \left[ \alpha \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right) \right] + M_{sl}^d + \sum_{s=1}^{s} D_{sl}^d \left[ u_i^l - u_i^s \right]. \tag{2.3}
\]

Additional phase-specific properties are indicated for the density, viscosity, pressure, and velocity, presented as \( \tilde{\rho}^i, \mu^i, p^s, u_i^s \), respectively. For each species, a new momentum equation arises. Coupling between each species occurs through source terms that model interfacial drag, \( D_{sl}^d \), and nondrag, \( M_{sl}^d \), effects. The nondrag effects include factors such as bubble-induced lift, virtual mass, and turbulent dispersion. The modeling of \( D_{sl}^d \) and \( M_{sl}^d \) are analogous to turbulence models, in that they are arrived at via an averaging process of the unresolved, multiphase interactions. When assuming a species-specific temperature, a similar process yields \( n \) energy equations. These equations are valid if the modeled, intercomponent-coupling is correct. In this methodology, with an arbitrary number of species, \( n \), the number of governing partial differential equations grows from \( 4+n \) to \( 5n \).

The basis of the homogenous-mixture assumption is that the velocity, pressure, and temperature are uniform amongst all species. There are two conditions where this is
valid. First, when the phases are separated and each phase is fully resolved. Secondly, conditions when the velocities of all species are uniform. In these applications, this case is of concern when considering air bubbles carried by water. Here, the velocities of the fluids tend to be uniform when the drag interactions, or the $D^{kl}$ terms, are strong with respect to the mass of the bubbles. Thus, a slip velocity between the fluids yields a drag that tends to recover a state with uniform velocities. Furthermore, a strong virtual mass term tends to dampen both the deviation and recovery of this uniform velocity. The remaining nondrag terms, $M^{kl}$, i.e. turbulent dispersion and lift, must also be relatively small compared to the drag to prevent other forces that can generate slip velocities. Temperature and pressure differences are also likely to occur, but the effects are relatively weak in large-scale cavitating flows. Explicitly, the homogenous assumption implies that

$$u_i = \sum_{s=1}^{n} \alpha^s u_i^s \approx u_i^s \text{ for all } k,$$

$$p = \sum_{s=1}^{n} \alpha^s p^s \approx p^s \text{ for all } k,$$

and $T = \sum_{s=1}^{n} Y^r T^s \approx T^s \text{ for all } k$.

This assumption results in a significant reduction of the number of governing equations, by recovering the continuum-based governing equations at the cost of removing the ability to account for subgrid-scale interactions. In cases where each component is fully resolved, or when the velocity vectors, pressures, and temperatures of each fluid are roughly the same, the assumption remains valid. For example, in sheet-type-cavitating and free-surface flows, the phases are separated where each component can be practically
resolved and the homogenous assumption becomes valid. However, in certain regions, such as bubbly mixtures, this may not be the case. Cloud cavitation and breaking waves have small-scale bubbles that cannot be resolved in practical applications. In these cases, the bubble momentum is low, indicating that water currents will have a strong impact on the bubble motions. The benefit of the homogenous assumption is reiterated, where the number of governing equations reduces from $5n$ to $4+n$.

A more detailed investigation of the homogenous-mixture assumption is performed using a single bubble and is related to the applications considered in these studies. First, assume that the temperature and pressure of the bubble are the same as the carrier liquid. In certain cases, only small deviations from this assumption are expected to occur. Examples are mass-transfer induced temperature changes or surface tension maintaining a pressure difference. However, the overall effects of these assumptions on hydrodynamic predictions are presumed to be negligible. This is not necessarily the case for velocity homogeneity. Consider the main forces on a single spherical bubble. A range of flows can be estimated using the drag and buoyant forces, which are the most significant forces on a bubble. Force relations for the drag and buoyant forces on a single sphere, with variations in the flow characteristics, are $F_D = \frac{1}{2} \rho L A V^2 C_D$ and $F_B = \frac{4}{3} \pi r^3 (\rho_G - \rho_L) g$, respectively. Maintaining $Fr$ number similarity, the ratio of the forces due to drag and buoyancy are expressed as

$$\frac{F_{D/B}}{F_{BUOYANT}} = \frac{3 Fr_s^2 C_D}{4 \left(1 - \rho_L/\rho_G\right)}.$$
where $Fr_B$ is the Froude number based on the bubble diameter and slip velocity between the bubble and mean flow. Note that in this analysis, other equivalent accelerations can readily substitute gravity. $C_D$ is the bubble drag coefficient, which is assumed to behave similar to that modeled in Kunz et al. [52]. Note that other neglected forces and modifications exist for multiple bubbles.

Results from Eq. 2.5 are plotted for various values of $Re_B$ and $Fr_B$ in Fig. 2-1. High $F_{D/B}$ values imply that the drag dominates the bubble motion and forces them to align with the liquid. Thus, the homogeneous mixture assumption becomes valid. At low $F_{D/B}$ values, the bubbles must be resolved or modeled with a slip velocity. Classifications of various flows with respect to this plot are noted in Fig. 2-1. In general, features to the right of Fig. 2-1 include large-scale cavities that are resolvable. The upper left region may include high velocity mixtures at a turbulent interface, the immediate wake of a cavity, cloud cavitation, etc. In these regions, the bubbles are dominated by drag and can be treated with a homogenous-mixture assumption. The problematic region for the homogenous-velocity assumption is in the lower left quadrant. These features are expected to occur in boundary layers, dispersed wakes, and lingering bubbles in a wake. All of these regions exhibit a range of velocity fluctuations, scales, and bubble sizes, that includes regions where the homogenous-velocity assumption becomes inappropriate.
The more problematic regime for the assumed velocity behavior occurs for droplets in a carrier gas. Using a similar analysis as used for a bubble, but for a water droplet by reversing $\rho_G$ and $\rho_L$, the contours plotted in Fig. 2-1 are shifted upwards by roughly 1.5 contour lines. This shift indicates a significant reduction in the drag force with respect to the gravity force. A more important feature is that droplets have a significant mass compared to air as a carrier fluid; the response time to equalize the drag forces is increased significantly (acceleration decreases by $1000^2$) as a result of this mass increase. This implies that significant slip velocities will exist for a droplet and that the homogenous-velocity assumption is not typically adequate. Fortunately, cavities are relatively unaffected from the droplets that form inside the cavities. The more important
features regarding cavity dynamics are turbulence, jet-cavity interactions, and large-scale reentrant water jets of which can destabilize the cavity interface [4].

2.1.2 Turbulence Modeling

Models turbulence for multiphase flows are based on standard single-phase methods. For the homogenous assumption and using mixture properties, the turbulence model is identical to those of single-phase models. In this work, a two-equation turbulence closure is used, and is given as

\[ \frac{\partial Q_T}{\partial t} + \frac{\partial F_j^T}{\partial x_j} = \text{PROD} - \text{DEST} \]  

The vector, \( Q_T \), includes the turbulent kinetic energy, \( k \), and dissipation rate, \( \varepsilon \), defined as

\[ Q_T = \begin{bmatrix} \rho k \\ \rho \varepsilon \end{bmatrix} \]  

The turbulent flux vector, \( F_j^T \), is given by

\[ F_j^T = \begin{bmatrix} \rho ku_j - \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \\ \rho \varepsilon u_j - \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \end{bmatrix} \]  

Finally, the turbulence production, \( \text{PROD} \), and destruction, \( \text{DEST} \), are given as

\[ \text{PROD} = \begin{bmatrix} p \\ C_p \rho \frac{\varepsilon}{k} \end{bmatrix} \]  

and

\[ \text{DEST} = \begin{bmatrix} \rho e F_{\text{DES}} \\ C_2 \rho \frac{\varepsilon^2}{k} \end{bmatrix} \]
Within the scope of this work, adequate transition behavior of the boundary layers is not modeled.

In the context of the $k$-$\varepsilon$ model, a DES variant is obtained by modifying the turbulent-energy destruction using the $F_{DES}$ the $DEST$ term in Eq. 1.18 [53]. Otherwise, $F_{DES}$ is unity and does not affect the model. This modification is given as

$$
F_{DES} \equiv \text{MAX} \left[ \frac{L_{\|}}{C_{DES} \Delta}, 1 \right],
$$

$$
\Delta \equiv \text{MAX}(\delta_x, \delta_y, \delta_z),
$$

and

$$
L_{\|} \equiv \frac{k^{3/2}}{\varepsilon}.
$$

With this rather subtle modification, a hybrid RANS/LES turbulence model is encapsulated. In regions where the grid resolution is coarse relative to the turbulent scales, near walls for example, $F_{DES} \rightarrow 1$ and the turbulence model behaves in the baseline manner. Where the grid resolution can support the large-scale eddies, the values $F_{DES}$ extend past unity and tends to amplify the destruction of $k$, thus, the modeling of $\mu_t$ reverts to model subgrid-scale stresses. With mesh refinement, this formulation is argued to behave similar to a Smagorinski subgrid-scale stress model [53]. Other suggested modifications [53], such as hybridized schemes between upwind-biased and central are doubtfully usable in such flows; this is because of the large density ratios combine with compressible effects occurring in these regions are likely to result in an unstable discretization method. For this reason, a $3^{rd}$-order, upwind-biased scheme is consistently used throughout the LES and RANS regions.

Eddy-resolving simulations imply highly a refined mesh and a small time-step sizes in order to resolve the larger turbulent eddies in the fluid flow. In DES simulations,
Courant numbers \((u\Delta t/\Delta x)\) less than unity and a nearly isotropic mesh are required to maximize accurately in the LES regions. The time-step size requirement is based on resolution and numerical accuracy arguments. The isotropic cell requirement prevents the cell-length scale, \(\Delta\), from reducing to the longest cell dimension. Furthermore, physical arguments imply that the LES regions need an isotropic mesh to prevent filtering turbulent scales in the energy cascade that can cause aliasing at the lower wave numbers \([54]\). In this work, the meshes only deviate slightly from these rules.

### 2.1.3 Cavitation Modeling

The transfer of mass between liquid and vapor is modeled with a finite-rate model cavitation model \([24]\) and is given by

\[
\dot{m}^- = C_{\text{det}} \hat{\rho}^v \alpha^l \frac{\text{MIN}[0, p - p_v]}{\frac{1}{2} \rho U^2 \tau},
\]

and

\[
\dot{m}^+ = C_{\text{prod}} \hat{\rho}^v \left(1 - \alpha^v\right)^2 \alpha^v \frac{1}{\tau}.
\]

These are introduced as source terms into the species-mass conservation equations, in Eq. 2.1, via the definition of \(\dot{\omega}^l\). The source term for the vaporous phase becomes, 
\(\dot{\omega}^r = -(\dot{m}^+ + \dot{m}^-)\), whereas an opposite rate must be included for the liquid phase as 
\(\dot{\omega}^l = + (\dot{m}^+ + \dot{m}^-)\). The model assumes that there are an infinite number of nucleation sites, which is a valid assumption for large-scale cavitating flows. In the liquid-vapor mass-transfer model, the destruction of liquid is related to the difference of the local pressure, \(p\), and vapor saturation pressure, \(p_v\). The production of liquid is based on the
interpretation of the work of Hohenberg and Halperin [55]. Within the context of Eq. 2.12, the saturation vapor pressure is an assumed function of temperature. The state of each phase is determined through the local, mixture temperature and pressure. Therefore, thermal effects from phase change are included through the absolute energies and densities associated with each pure phase.

An alternative approach to modeling the phase change in cavitating problems is using a single-phase formulation with a uniform state equation that is valid for all phases. This includes validity for the compressed, superheated, and saturated mixture of the water. A common approach is using a barotropic-state equation, for example see Coutier-Delgosha [56], which is equally as effective as the finite-rate mass transfer model. However, the formulation presented in this work is preferred due to the added flexibility of easily introducing additional noncondensable species. Such a capability enables free surface and ventilated cavitation predictions with little to no additional effort.

2.1.4 Homogenous Mixing Rules

Homogenous mixing rules need to be defined to couple the transported species to the mixture-mass, momentum, and energy equations. The reader is reminded that the superscript \( s \) enumerates the gaseous species and that the tilde above the density indicates that it is defined by the rule of Amagat. The mixture density is given by the volume-averaged density, or

\[
\rho = \sum_{\text{vol}}^{n} \alpha^* \tilde{\rho}^s. \tag{2.14}
\]
The viscous stress tensor takes on the usual form as

$$t_{ij} = \mu_{m,i} \left( u_{i,j} + u_{j,i} - \delta_{i,j} \frac{2}{3} u_{k,k} \right). \quad 2.15$$

The mixture viscosity is also defined based on volume averaging, which is given by Eq. 2.16. The modeled turbulent viscosity is computed using Eq. 2.17 and the overall modeled viscosity is given in Eq. 2.18.

$$\mu_m = \sum_{i=1}^{n} \alpha_i \mu_i. \quad 2.16$$

$$\mu_t = \frac{\rho C_p k^2}{\epsilon}. \quad 2.17$$

$$\mu_{m,t} = \mu_m + \mu_t. \quad 2.18$$

Note that the subscript, $m$, indicates a mixture quantity. The mixture Prandtl number, $Pr_m$, is also computed using a local volume average, using

$$Pr_m = \sum_{i=1}^{n} \alpha_i Pr_i. \quad 2.19$$

Whereas the mixture specific heat, an extensive property, uses a mass-fraction averaging, or

$$C_{p,m} = \sum_{i=1}^{n} Y_i C_{p,i}. \quad 2.20$$

Finally, the turbulent-mixture thermal conductivity is computed as

$$\kappa_{m,t} = \left( \frac{\mu_m}{Pr_m} + \frac{\mu_t}{Pr_t} \right) C_{p,m}. \quad 2.21$$
2.2 Numerical Description

The computations performed in this thesis employ the finite-volume code, UNCLE-M. This is a parallel, block-structured solver that originates from the UNCLE CFD code [57]. The capability of UNCLE-M has been extended to handle a broader range of problems than the original UNCLE version; this includes an \( n \)-species multiphase capability (with mass transfer) as well as compressible-fluid flows. The general \( n \)-species conservation equations are formulated using homogeneous-mixing rules that include the flexibility to incorporate any two-variable equation of state for any of the species. Thus, the mixture state is a function of the volume fraction as well as the mixture pressure and temperature. This particular fully compressible formulation uses a preconditioning approach that enables the efficient simulation at all-Mach numbers [58]. Two-equation mixture-based turbulence models are implemented and include a DES capability [59]. An implicit, dual-time, second-order-accurate integration technique is used with a formal third-order spatial accuracy. This formulation facilitates the necessary physics for the varied Mach number regimes required for the problems of interest. The methods used in this formulation are discussed in detail below.

The differential form of the preconditioned computational model is described by

\[
\frac{\partial Q_c}{\partial t} + \Gamma^\tau \frac{\partial Q}{\partial \tau} + \frac{\partial F_j}{\partial x_j} - \frac{\partial F^\tau_j}{\partial x_j} = H .
\]

2.22

This is a vector representation of the mass, momentum, energy, and species conservation equations. The corresponding conservative variables, \( Q_c \) (Eq. 2.23), primitive variables,
Referring to Eq. 2.22, the first elemental equation (row) represents the conservation of mass for the mixture. Note that forming a problem with $n$ species required $n$ scalar equations for mass conservation. Using this form with a mixture mass as the first conservation equation, replaces one species-mass conservation equation. The following two rows are the mixture-based momentum and energy equations. The last element represents the conservation equations for the individual gaseous species.

2.2.1 Preconditioning Approach

The term $\Gamma^\tau \partial Q / \partial \tau$, from Eq. 2.22, is a result of the Choi-Merkle dual-time preconditioned approach [60, 61]. This term is introduced to alleviate the natural decoupling between the physical momentum and continuity equations occurring in low-Mach number flows. The methodology of Venkateswaran and Merkle [62] is applied to derive a pseudo-inviscid eigensystem, which ideally dominates the convergence of the pseudo-time marching system. In the dual-time approach, a time-accurate solution is
recovered after driving the pseudo-time residuals to small values over each physical time step. The preconditioning matrix, $\Gamma^p$, used in UNCLE-M is given as

$$\Gamma^p = \Gamma^p_I \Gamma_a^Y,$$

where

$$\Gamma^p = \begin{bmatrix} 
\rho^p_r & 0_j & \rho^p_r & (\rho^p_r)^s \\
u_i \rho^p_p & \rho \delta_{ij} & u_i \rho^p_r & (u_i \rho^p_r)^s \\
h_i \rho^p_r + \rho h_p - 1 & \rho u_j & h_i \rho^p_r + \rho h_p & (h_i \rho^p_r + \rho h_p)^s \\
Y^p_i \rho^p_r & 0_j & Y^p_i \rho^p_r & (\rho \delta_{j} + Y^p_i \rho^p_r)^s 
\end{bmatrix}$$

and

$$\Gamma_a^Y = \begin{bmatrix} 
1 & 0_j & 0 & 0^s \\
0_i & \delta_{ij} & 0_j & 0_i \\
0 & 0 & 1 & 0 \\
(\gamma^p_r)^s & (0)^s & (\gamma^s_r)^s & \left(\frac{Y^p_i (\dot{\rho^p} - \dot{\rho}) + \ddot{\rho} \delta_{j}^s}{\rho}\right) 
\end{bmatrix}.$$

The superscripts $r$ and $s$ indicate column and row vectors, respectively, and operate over species $2, 3, \ldots, n$. The subscripts $i$ and $j$ correspond to column and row vectors, respectively, that represent the Cartesian direction. The subscripts of $T$, $Y$, and $P$ indicate partial derivatives of the term with respect to temperature, mass fraction, and pressure, respectively. The primed quantities are modified in the preconditioning approach; this artificially alters the modeled sound speed to yield a well-conditioned eigensystem. The primed values relate to the modeled sound speed, $a'$, by

$$\rho^p_r' = \frac{\partial \rho^p}{\partial p} = \frac{1}{a'^2}. \quad 2.29$$

The goal of preconditioning is to prevent the eigenvalues of the linear system from diverging in the low-Mach number limit. As previously discussed, the eigenvalues indicate speeds at which information propagates. In the one-dimensional case they are $u+a$, $u-a$, and $u$. At low Mach numbers the magnitude of $u+a$ becomes large with respect to $u$, which are eigenvalues of the formed linear system. When these eigenvalues are
disparate, the formed linear system becomes poorly conditioned and difficult to solve. Specifically, solutions to these poorly conditioned systems are inaccurate and slow to converge. By artificially modifying the sound speed in pseudo time, or the pseudo-sound speed, a well-conditioned system is achieved. In application, the pseudo-sound speed is cut off as

\[ a'^2 = \min\left( \max\left( \beta, V_{\text{Local}}^2 \right), a^2 \right), \]

where \( \beta \) is the maximum value of the pseudo-sound speed squared. The typically \( \beta \) value is roughly \( (10V_\infty)^2 \). Such a modification alters the eigenvalues of the linear system, that is for a one-dimensional problem, to \( u, u+a', \) and \( u-a' \). Thus, when needed, the sound speed in pseudo time does not grow large with respect to the local velocity and the linear system is effectively conditioned. In the dual time formulation, only the pseudo-time derivatives are affected. Thus, upon convergence, the underlying governing equations are recovered. Further details of the approach are available in the work of Kunz et al. [24] or Potsdam et al. [63].

### 2.2.2 Numerical Solution Procedure

As a finite-volume approach is used, the spatial differencing is based on the numerical flux computed at the cell faces. Several schemes are available to compute this flux within UNCLE-M, which are based on either flux-vector or flux-difference splitting.

**Inviscid Flux Formulation:**
For flux-vector splitting the AUSM-based schemes are used, specifically LDFSS [34] and AUSM-\textsuperscript{+}up [35] (reviewed in Section 1.3.1.2). In the development of the AUSM schemes the flux vector is split into an advective and pressure part as

\[ F_j = \begin{bmatrix} \rho u_j \\ \rho u_j u_j + p \delta_j \\ \tilde{p}_j \alpha_k u_j \end{bmatrix} = F_j^a + F_j^p = \begin{bmatrix} \rho \\ \rho u_i \\ e + p \end{bmatrix} + \begin{bmatrix} 0 \\ p \\ \delta_j \end{bmatrix}. \quad (2.31) \]

After this splitting, both fluxes are evaluated using upwind-based polynomials of the local Mach number. Variations of these polynomial functions are the main differences between the various AUSM-type schemes [34, 35].

A flux-difference splitting scheme based on Roe’s method is also used [36]. Recall from Chapter 1, for flux difference split cell-centered finite volume schemes the Roe flux on the cell face is constructed as

\[
\tilde{F}_j \left( i + \frac{1}{2} \right) = \frac{1}{2} \left( F_j (i) + F_j (i+1) \right) - \frac{1}{2} \left[ \overline{A}_{i+1/2} \right] \left( Q_R \left( i + \frac{1}{2} \right) - Q_L \left( i + \frac{1}{2} \right) \right). \quad (2.32)
\]

The first term on the right-hand side is computed using a linear approximation (central differencing), while the second term can be interpreted as a matrix dissipation. The flux-vector Jacobian matrix, \( \overline{A}_{i+1/2} \), is defined as \( \frac{\partial F_j}{\partial Q_{i+1/2}} \). In low-Mach number flows, the value of \( \overline{A}_{i+1/2} \) can be computed using arithmetic averaging rather than true Roe averaging [36]. This is reasonable for moderate temperature and pressure jumps across a shock or interface. The higher-order cell-face primitive-variable interpolations are computed using a third-order-accurate upwind-biased van-Albada flux-limited MUSCL
formulation [64]. Note that this solution technique captures interfaces based on the shock-capturing method as described in Chapter 1.

**Temporal Integration:**

A fully coupled, implicit numerical approach is used to solve the resulting linear system. In delta form, this linear system becomes

\[
\left[ \frac{\Gamma^p}{\Delta \tau} + \frac{\partial Q_c}{\partial Q} \frac{3}{2\Delta t} + \frac{\partial Q_c}{\partial Q} \left( \frac{\partial}{\partial x_j} A_j \right) + \frac{\partial}{\partial x_j} \left( R \frac{\partial}{\partial x} \right)_j \right] \Delta Q = -R^m \quad 2.33
\]

Here, the flux-vector Jacobian, \( A_j \), is defined with respect to the conservative variables, or 

\[ A_j = \frac{\partial F^m}{\partial Q_c}. \]

The matrix \( R_j \) includes coefficients of the viscous dissipation terms as well as the artificial-dissipation Jacobian. The residual vector, \( R^m \), at a point \( m \) in pseudo time, is defined as

\[
R^m = \frac{3Q^n - 4Q^n + Q^{n-1}}{2\Delta t} + \left[ \frac{\partial F_j}{\partial x_j} - \frac{\partial F^v_j}{\partial x_j} - H \right]^m, \quad 2.34
\]

and the solutions subscripted by \( n \) and \( n-1 \) correspond to the previous two time levels. The time derivative in this particular form in the residual vector, \( R^m \), yields a formally second-order accurate temporal scheme. The implicit integration in pseudo time uses a block-symmetric, Gauss-Seidel iteration technique to solve the resulting linear system. In the numerical solutions presented here, the inviscid-flux Jacobians, \( A_j \), are formed using either numerical or analytical methods. The numerically determined Jacobians are formed by tracking the response of \( F_j \) with perturbed \( Q_c \) values, while the analytical form uses exact derivatives. The resulting numerical performance seems problem dependent and no comparison is presented here, however, the analytic form requires fewer arithmetic
operations. The viscous-flux Jacobians are always computed numerically. Note that the turbulence-model equations are solved in a segregated fashion, where the linear systems resulting from Eq. 2.22 and Eq. 2.6 are not coupled.

**Application of Boundary Conditions:**

The boundary conditions are applied using ghost cells. The number of ghost cells required depends on the number of cells in the stencil of the interpolation scheme. Within the boundary conditions, the properties of the primitive-variable behavior are specified or fixed. Of course, each type of boundary condition has a slightly different approach. At the far-field boundaries, the boundary condition behavior depends on the flow direction with respect the cell face. First, consider a boundary condition at a face, $i+1/2$, that falls between cells, $i-1$ and $i$, where the cell at $i-1$ is in the domain and $i$ is the first ghost cell. When the fluid is moving out of the domain, an outflow boundary conditions is applied such that

$$
\begin{bmatrix}
p \\
u_j \\
T \\
\alpha'_{ij}
\end{bmatrix} =
\begin{bmatrix} 2p_{\infty} \\
-2u_j \\
2T \\
0
\end{bmatrix} +
\begin{bmatrix} -p \\
-u_j \\
-T \\
-\alpha'_{i-1}
\end{bmatrix} +
\begin{bmatrix} 0 \\
-\alpha'_{i-2}
\end{bmatrix}.
$$

2.35

These conditions specify the pressure, $p_{\infty}$, at the outlet face and extrapolate the other variables from within the computational domain. The inflow boundary conditions are slightly different in that the ghost cells are two cells deep resulting from the upwind-bias scheme. Similarly, the values of the primitive variables are applied, where this time the inflow cell face is positioned between the $i-1$ and $i$ cells where $i$ is inside the domain. The two ghost cells are specified as
This is opposite to the outflow boundary case, where here the pressure is extrapolated from the computational domain and the velocities, temperature, and volume-fraction values of each species are specified.

In general, the walls are assumed to be adiabatic surfaces with no mass flux through the surface. Again, two ghost-cell layers are used at the walls. In this example, $i$ is inside the computational domain and the boundary conditions are applied as

$$
\begin{bmatrix}
    p \\
    u_j \\
    T \\
    \alpha'^i_{j-1}
\end{bmatrix}_{i-1} = \begin{bmatrix}
    0 \\
    u_j \infty \\
    T_\infty \\
    \alpha'_{\infty}
\end{bmatrix} + \begin{bmatrix}
    2p \\
    0 \\
    0 \\
    0
\end{bmatrix} - \begin{bmatrix}
    p \\
    u_j \\
    T \\
    \alpha'^i_{j+1}
\end{bmatrix}_{j+1}
$$

and

$$
\begin{bmatrix}
    p \\
    u_j \\
    T \\
    \alpha'^i_{j-2}
\end{bmatrix}_{j-2} = 2 \begin{bmatrix}
    p \\
    u_j \\
    T \\
    \alpha'^i_{j-1}
\end{bmatrix} - \begin{bmatrix}
    p \\
    u_j \\
    T \\
    \alpha'^i_{j+1}
\end{bmatrix}
$$

Here, the pressure, volume fraction, and temperature gradients are all forced to zero, whereas the velocity is forced to the wall velocity, $u_{j,w}$.

Several variations of the described boundary conditions exist. A supersonic form of the inlet and outlet exists that takes into account the characteristics in applying the boundary conditions. Furthermore, inviscid walls and symmetry planes also exist. These cases are essentially identical to each other and are similar to the wall, only differing in that they do not enforce a no-slip condition. All of the boundary conditions are applied in the formation of the linear system and residual vectors, but are not incorporated into the linear system. The boundary conditions are applied by updating the primitive variables just after each pseudo-time step.
2.2.3 Advancement of Baseline

A significant portion of the numerical solution scheme within UNCLE-M was established prior to this work. Advancements specific to this work are summarized in the following section.

Overset-Grid Capability

An overset grid capability was implemented in UNCLE-M using DiRTLib [65]. Note that a more detailed description of the numerical formulations is provided in Chapter 5. DiRTLib is a library of subroutines that performs the necessary data passing and interpolation of the flow variables from one domain to another. For an overlapped grid, interpolations to the fringe cells occur between neighboring donor-cells that exist within another mesh. The domain connectivity information includes the indices of the donor cells and the interpolation weights. This information is computed through SUGGAR [66], which also provides hole cutting and fringe-cell information. The overset method is valid for both static and dynamic cases.

Multiple-Dynamic-Body Capability

Using the overset method, an existing single-body motion capability was extended to handle multiple bodies in relative motion. This motion can be determined from the computed flow field and use the rigid-body equations of motion with up to six degrees of freedom, be prescribed, or use a combination of both. The free-body motion incorporates the force and moment integration from the flow solution on the solid surfaces of the rigid bodies. When using the overset method, multiple grids may be present on a common physical space on the surface and need to be accounted for when
integrating the force and moments. Otherwise, the force and moment integration values will become erroneous by accounting for the same space on a physical surface multiple times. Such a problem is addressed using USURP [67], which is a tool that assigns integration weights to each cell face that is on a physical surface. The weights given are bound between zero and unity, locally sum to one, and are computed based on mesh quality. Using the integration weights provided by USURP effectively eliminate such issues in the force and moment integration.

Residual Smoothing

In several cases, the improvement in the algorithm stability, for both explicit and implicit pseudo-time integration, is achieved using the implemented residual smoothing techniques of Jameson [68]. Residual smoothing is often used to improve the stability characteristics and convergence of explicit-based steady-compressible-flow schemes. In the approach, spatial smoothing of the residual vector, $R^m$, is performed using a weighted mix of the unsmoothed residual and the Laplacian of $R^m$. The explicit version of the smoothing procedure in terms of Eq. 2.33 is given by

$$R^n = R^{m^*} + \epsilon \nabla^2 R^{m^*},$$

where $R^{m^*}$ is defined by Eq. 1.2. Although residual smoothing is typically used for steady state problems, in the dual-time formulation it is introduced as an additional artificial source. This term physically appears as another source that diminishes as the system converges in pseudo time.

Adaptive Free-Surface Meshing Capability
An adaptive mesh method that is suitable for free-surface flows has been developed by the author and introduced into UNCLE-M. This method is described in more detail in Chapter 5. It is based on a combined version of surface-tracking and volume-fraction approaches. Such an approach eliminates the restrictions of a surface-tracking approach, while improving efficiency of the cell usage. The method is well suited for free-surface problems using structured, unstructured, and overset grids.

*Level-Set Based Sharpening*

A level-set approach, reformulated by the author in volume-fraction-based variables is implemented in UNCLE-M. The method is described in detail in Chapter 5. The approach is valid for any number of fluids and is cheaper than previous methods.

*Various Equations of State*

As indicated above, UNCLE-M accommodates arbitrary state relations. Several additional equations of state have been implemented into UNCLE-M within this work. This includes the addition of the Peng-Robinson equation of state, a linearized-equation of state that is well suited for multiphase simulations, and saturation curves for various cryogenic fluids.

2.2.4 Computational Grid Generation

Most of the structured grids used to obtain solutions presented here were generated using Chimera Grid Tools [69], an overset grid-generation package developed at NASA. The grids, in general, follow guidelines based on the “best practices in overset grid generation” [70]. In particular, exceptions to those guidelines include larger wall
spacing, as here a high-Reynolds number spacing of $y^+ \approx 100$ is used, which is the appropriate value when using log-law wall functions. Furthermore, regarding wall spacing in the multiphase problems of interest, estimating the wall $y^+$ values is not straightforward. This is a result of large variations in the kinematic viscosity occurring in gas liquid systems. Additional mesh refinements are required for the modeling of cavitation as studied in Chapter 4 for the problems of interest.

### 2.3 Summary of Modeling Approach

The presented numerical scheme aligns well with cavitating vehicle design needs. This includes the physical modeling requirements, where the formulation handles varied equations of state for $n$-species flows. Such a formulation is useful for simulating rocket propulsion plumes, various substances, etc. This is integrated with modern turbulence model formulations that simulate turbulence and can yield turbulence-induced loads without very large mesh sizes. Finally, with the overset-mesh approach, dynamic and 6DOF body-motion simulation capabilities exist. In total, the aforementioned capabilities and improvements result in a useful analysis tool for real-world design.
Chapter 3

Validation Case Description

Several relevant test cases are used to evaluate and illustrate the potential of the numerical scheme. Short descriptions of these test cases are provided in the following chapter are ultimately used to evaluate the capability of the scheme. These cases are appropriately categorized by their specific measures of validation for artificial supercavitation, vaporous cavitation, and free-surface flows.

3.1 Test Cases

The validation cases are based on documented experiments or semi-empirical theories. These test cases will be referenced throughout these validation studies and in the remaining work.

3.2 Artificial Cavitation Validation Cases

The chosen cases for artificially-cavitating flows span a range of cavity types and comprise, in an isolated sense, many of the parts involved in a supercavitating vehicle. These examined cases include a disk cavitator at a high Froude number, a lifting cone cavitator, a supercavitating body, and a lifting surface piercing a cavity. These cases validate the baseline numerical scheme and meshing strategies.
3.2.1 Test Case AC1: Disk Cavitor

A hypothetical ventilated disk cavitor, displayed in Fig. 3-1 below, is used as baseline test case. To reduce the computational time, gravity is assumed to be negligible so that only an axisymmetric slice of the geometry needs to be modeled. This case is convenient as it remains relevant, while greatly reducing the computational effort. Furthermore, semi-empirical relations, discussed in Section 1.3.3, can be used to validate the cavity-profile predictions. These semi-empirical profiles predict experimentally measured cavity profiles accurately for disk cavitors at high values of $Fr_n$ [4]. The assumed conditions for this test case are: $Fr_n=\infty$, $Re_N=90\times10^6$, and $C_Q=0.5$. In Fig. 3-1 (a), the general arrangement of the case, an isolated disk cavitor that is ventilated to form an artificial supercavity. In Fig. 3-1 (b), the medium-resolution mesh is shown along with the volume fraction of air (air=red, water=blue). The cavitor is modeled as an infinitesimally thin disk cavitor at the upstream most part of the red colored cavity.

![3d View](image1.png) ![side view](image2.png)

Figure 3-1: Axisymmetric disk cavitor ($6^\circ$ wedge simulated) ventilated on the aft faces of the cavitor. The conditions are $Re_N=90\times10^6$, $Fr_N=\infty$, $C_Q=0.5$. The isometric view shown in part (a) has a cavitor colored by pressure and the cavity shape ($\alpha=0.5$ isosurface) shown in blue. The side view and the medium-resolution computational mesh are shown in part (b), where the contours are colored by the volume fraction (red=air, blue = water).
3.2.2 Test Case AC2: Artificially-Supercavitating Cone Cavitation

To explore the ability to predict cavitation hydrodynamics, a 15°-half-angle, cone-shaped cavitation based on a summary of experiments by Kiceniuk [71]. The cavitation is set to a lifting configuration, relevant for trimming or maneuvering a vehicle. Based on the actual diameter, rather than the commonly used effective-disk diameter, the corresponding conditions are: \( Re_N=3.2 \times 10^7 \), \( Fr_N=72.0 \), and \( C_Q=1.0 \). Note that these supercavities are typically insensitive to the Reynolds number; so comparisons to experiments remain valid. The computational mesh is displayed in Fig. 3-2, which uses 1.28×10^6 computational cells. A pseudo body sits in the cavity where, based on theory related to supercavities, this should not affect the predicted hydrodynamics unless it pierces the cavity. Note that no cavity-piercing support strut was modeled for this case.
3.2.3 Test Case AC3: Artificially-Supercavitating Body

An artificially-supercavitating body with significant cavity-body interactions is examined based on experiments conducted in the University of Minnesota, St. Anthony Falls Laboratory’s high-speed water tunnel [72, 6]. The simulated flow conditions correspond to $Re_N$ of 56,000 and $Fr_N=26.7$, and a range of $C_Q$ values is simulated. The experiments display a Gilbarg-Efros-type closure, where the cavity terminates with a reentrant jet onto the body [4]. This differs from pure twin-vortex or reentrant closure, as cavity-body interactions become important. The test section dimensions are 19cm×19cm.
and the body geometry, displayed in Fig. 3-3, has a 1 cm cavitator diameter and is roughly 11 cm long. The body is supported in the center of the tunnel using a streamlined, elliptical strut and is displayed in Fig. 3-3 (b).

![Figure 3-3](image)

(a) Geometry (b) Photograph of experiment

Figure 3-3: Experimental setup description of the geometry (a) and a photograph displaying cavity, body, and support strut. These figures are borrowed from Schauer [72].

The CFD model is designed to represent the experiment. The water-tunnel walls are treated as inviscid, and in most results, the strut assembly is ignored. The geometry of the body is modeled as specified by Schauer [72]. The mesh that assumes that a plane of lateral symmetry exists uses approximately $1 \times 10^6$ computational cells, which is presented in Fig. 3-4. (a). The full three-dimensional model uses a mirror image of the mesh from the lateral-symmetry case to make a full-three dimensional domain. The support strut is incorporated using an overlapping O-grid. In all of these cases, the axial spacing is roughly $0.025D_N$, and is sustained from the separation point, off the cavitator, through to the immediate wake region. The wake-resolving mesh uses a nearly isotropic grid, while the cells conforming to the body maintain a 3:1 azimuthal to axial spacing. Thus, the wake resolution is a “proper” DES-quality grid, but near the body, the grid resolution is slightly anisotropic. Regardless, the ability to simulate large scale turbulent structures within the cavity is still revealed.
3.3 Vaporous Cavitation Validation Cases

Several vaporous cavitation test cases are used for the validation of the vaporous cavitation flows. These investigations include studies of several, quite complex, geometries and state relations.

3.3.1 Test Case VC1: Twisted Hydrofoil

The geometry examined is a twisted-pitch wing profile positioned in a water tunnel based on the experiments of Foeth et al. [73] and Foeth [74]. A diagram of the
wing, in Fig. 3-5, displays the span-wise twist ranging from 0 degrees at the root, to 11 degrees at the mid-span. This static case is intended to capture flow features similar to those in a rotating propeller bladed [73]. The sectional profile is a NACA 0009 with a slightly blunted trailing edge. The modeled portion of the water tunnel extends for 7 chord lengths in the axial direction, 2c upstream of the wing leading edge, and 4c downstream of the trailing edge and is displayed in Fig. 3-5 (b). The tunnels walls, shown in blue, are modeled as inviscid and represent a 0.3m×0.3m test section. Note that all pitch rotations are based at the point where the wing is affixed to the tunnel wall, with rotations about the leading edge. A lateral plane of symmetry is assumed at the center plane of the tunnel. Although the $Fr_c$ is high, gravity is included in the negative-z direction. The red boundaries indicate the inflow and outflow planes. Note that the corresponding vapor pressure is specified as a cavitation number relative to the outflow pressure and inflow velocity, or $p_v=p_b-0.5\rho_\infty V_\infty^2\sigma_v$.\[\text{\textcopyright 2022}\]
The computational mesh used throughout these studies is displayed in Fig. 3-5 (c-d). An overset-grid system with overlap minimization that prefers retaining the higher quality cells is used. The wing-conforming O-grid is displayed in red, while the tunnel-resolving grid is displayed in blue. The meshes are made in accordance to "best overset grid generation practices" [70], implying that $\Delta s$ at the leading edge is 0.001$c$, at the trailing edge is 0.002$c$, and spacing ratios of 1.1 are maintained. Adjustments to these guidelines include a wall spacing near a $y^+$ of a 100 (a benefit of using wall functions).
and refinements to accommodate cavity and turbulent flow features. In the cavitating regions, a more refined mesh is use. This refinement includes a LES-appropriate grid, implying a nearly isotropic mesh to improve the simulation of the cavity dynamics associated with the turbulent-scale motions.

### 3.3.2 Test Case VC2: Cryogenic Cavitating Hydrofoil

A cavitating hydrofoil in a cryogenic flow, based on experiments from Hord [75], is examined. This enhances the validation from the former, twisted hydrofoil, case as the vapor pressure exhibits strong temperature dependencies. The hydrofoil is pitched at $\alpha=0$, has no camber, uses a leading edge radius of 0.396cm, and has a chord length of 6.35 cm. Note that the ratio of the hydrofoil thickness to wall height is 0.32 suggesting strong tunnel-wall-hydrofoil interactions. In fact, these interactions are exacerbated while cavitating where the foil and cavity occupy as much as half of the test section, which is displayed in the experimental photograph in Fig. 3-6. The general setup of the experiment can also be observed in the figure. Note that both the cavity and hydrofoil show as black on the photograph, making it difficult to make out the hydrofoil surface from this picture. Additional details of the hydrofoil profile are available in Hord [75]. The specific case examined herein is 290C, as listed in the report of Hord [75], which corresponds to the hydrofoil in liquid nitrogen at: $Re_c=9.1 \times 10^6$, $T_\infty=83.06K$, and $\sigma_\infty=1.7$. This particular case was chosen as past CFD studies have also investigated this case in detail [76].
The mesh used in this study is overlaid on the experimental photograph in Fig. 3-7. Note that the hydrofoil shape is now revealed with respect to the cavity. A two-dimensional mesh containing 30,000 cells is used. Several approximations of the geometry and physical characteristics are made. As the foil is not lifting, a plane of symmetry is assumed about the center of the tunnel. Downstream of the hydrofoil, where a complex sting mounting assembly exists, a tapered geometric recovery is assumed. The effects of these downstream assumptions are expected to be unimportant. The upstream is modeled to the point where the measurements of the free-stream temperature and pressures occur, which is shown in Fig. 3-7. The total properties are specified at the inflow, where the outlet pressure is altered until the conditions at the measurement location closely match those recorded experimentally. The tunnel-wall boundary layers are not modeled, as studies showed that the impact is minimal; an expected result for a high-Reynolds number flow. In this case, the hydrofoil boundary layers used a $y^+$ value of 1 at the wall to achieve adequately resolution of thermal boundary layer.
3.3.3 Test Case VC3: Oscillating, Cavitating Wing

An oscillating hydrofoil was tested in the 12” water tunnel at Penn State’s Applied Research Laboratory (PSU-ARL) [77]. The geometry modeled resembles a finite wing, or fin, and has an aspect ratio of 4.0. The cross section is an in-house, proprietary, ARL section is symmetric and 15-percent thick. The free-stream velocity corresponds to a chord Reynolds number of 5.0×10^5. The static pressure in the test section is adjusted to vary the cavitation number. The fin-pitch oscillations are controlled using a camshaft, where the motion is displayed in Fig. 3-8. Note that some high frequency pitch oscillations exist, but are smoothed for the prescribed motion in the CFD models. The normal force was measured using a force balance and then converted to a lift coefficient. The measured forces are displayed for different cavitation numbers in Fig. 3-8. Additionally, video was recorded throughout the oscillation. The pitch ranges from -2 to
11 degrees, and the cycle reduced frequency, \( k \), is about 0.0025. Unsteady effects are typically considered to be important for reduced frequencies greater than 0.05. Thus, this case is insensitive to unsteady effects, but displays a significant amount of stall behavior at lower cavitation numbers. These experiments are useful in order to validate quasi-steady CFD results, which can then be extended to the unsteady realm.

Two- and three-dimensional simulations of a moving control fin in a water-tunnel configuration are examined. The fin is examined in both a noncavitating (single-phase), and cavitating condition. In order to simulate the pitching motion in the tunnel, overset grids are used. The mesh used requires roughly \( 1.4 \times 10^6 \) cells. The general grid scheme is provided in Fig. 3-9 below. In Fig. 3-9 (b), an O grid that fits to the base of the fillet and is extruded through the depth of the fin to minimize the required hole cutting. It is difficult to see in the provided figures, but past the depth of the fin, through to the water

![Figure 3-8: Experimental measurements of the lift for a pitching hydrofoil with varied cavitation numbers.](image)
tunnel wall, an additional box grid is used. Also, in Fig. 3-9 (b), the hydrofoil is modeled using Chimera-style C-grids. The one on the suction surface uses a higher resolution, 179x40x62, grid and on the lower surface, the resolution is coarsened in the surface-normal direction, and the grid is 125x30x62. This is to take advantage of the hydrofoil pitch range from +11 degrees to –2 degrees, where cavitation is expected to be dominant on the suction surface. Thus, the upper surface uses the more spatially dense C-grid to better resolve and sharpen the large volume fraction gradients needed to represent the cavity. The lower surface uses a coarsened grid, closer to what is appropriate for high-Reynolds-number single-phase flows, and an overlapping region exists at the hydrofoil leading edge. A high-resolution Cartesian mesh is used downstream of the hydrofoil to maintain a high resolution in the wake and maintain vapor when present.

Figure 3-9: General overset-grid scheme used for the three-dimensional cases. The simulated walls are colored by pressure, and slices at the mid-span plane and through the trailing edge are shown in red and blue respectively. The remained tunnel walls are modeled with displacement thickness.
The three-dimensional cases modeled all water-tunnel boundary layers. The fin-mounted wall uses a wall-function-based method to model and resolve a turbulent boundary layer. The remaining tunnel-wall boundary layers are treated with inviscid boundary conditions, but are displaced by a polynomial approximation to the turbulent boundary-layer displacement thickness. This includes all of the important tunnel-wall effects, including any viscous interactions of the fin-wall junction.

3.4 Free-Surface Validation Cases

Relevant validation cases for free-surface flows are also discussed. These studies span from submerged-body-induced wave fields, to surface piercing devices. These two cases examine all the required physical characteristics of the flows of interest. That is submerged, cavitating bodies, base-vented supercavitation, and free-surface ships.

3.4.1 Test Case FS1: Submerged Hydrofoil

These studies are based on the experiments performed by Duncan [78], where the measured free-surface deflections due to presence of a submerged, NACA 0012, hydrofoil are available. The hydrofoil is positioned in a lifting configuration, \( \alpha = 5^\circ \), and is at \( Re_c = 1.45 \times 10^5 \), and \( Fr_c = 0.57 \). Several submergence depths are considered. Integral quantities of the hydrodynamics and wave characteristics such as the wave amplitude, damping, and length are used as metrics to examine the mesh-resolution requirements. The general, overset-based, computational mesh scheme is displayed in Fig. 3-10.
3.4.2 Test Case FS2: Surface-Piercing Hydrofoil

Experiments from a surface-piercing, NACA 0024 hydrofoil are used for further validation of free-surface flows relevant to bodies that pierce the free surface [79]. In this experiment, the hydrofoil is towed through a long tow tank using the device shown in Fig. 3-11 (a). Detailed measurements of the free surface are recorded throughout the towing run. This is a relevant validation case for hydrofoil struts, surface ships, or perhaps water-entry supercavitation problems. The experimental conditions are at $Fr_c=0.55$ and $Re_c=2.26\times10^6$. 

Figure 3-10: Overset grid system used to examine free-surface mesh requirements. The red grids remain static, while the blue (free-surface resolving) and black (water-field resolving) meshes are refined and coarsened. The black line indicates the free-surface location.
In this case, $1.83 \times 10^6$ cells are used to model the full three-dimensional experiment. The foil is modeled statically, with an oncoming velocity. For comparison purposes, with similar geometric simplifications as made by Kandasamy 	extit{et al.} [80], the grid could be reduced to roughly 400,000 cells while maintaining the similar grid density rules. Fig. 3-12 displays the computational mesh used. The entire tank is modeled with a single mesh. The hydrofoil-conforming mesh uses a C-grid and neglects the gap between the wing tip and the bottom of the tank. The hydrofoil is nonlifting so the tip effects should be small.

Figure 3-11: Description of experiments of a surface-piercing, NACA 0024 hydrofoil. Both the diagram and photograph are borrowed from Metcalf 	extit{et al.} [79].
Figure 3-12: Computational mesh used for the surface-piercing, NACA 0024 hydrofoil. Upper plot shows the entire domain, flow goes from right to left. In the lower plots, the flow goes from left to right, and in the left and right plots are the side and top views, respectively.
3.5 Summary of Test Cases

The test cases are categorized by artificial supercavitation, vaporous cavitation, and free surface flows. The ultimate goal of the validation effort is to develop confidence in each component of the overall problem, i.e. cavitating flows near free surfaces. Table 3-1 gives a brief summary of the previously discussed validation cases. Again, these test cases are based on documented experiments or semi-empirical theories, and are relevant for evaluating and illustrating the potential of the baseline and developed numerical schemes. Short descriptions of these test cases provided as a reference, and are categorized by artificial supercavitation, vaporous cavitation, and free-surface flows. These cases include most of the design-relevant aspects of the supercavitating vehicles of interest.

Table 3-1: Table summarizing the validation cases.

<table>
<thead>
<tr>
<th>Case</th>
<th>Description</th>
<th>Conditions</th>
<th>Significance of Validation</th>
</tr>
</thead>
<tbody>
<tr>
<td>AC1</td>
<td>Axisymmetric Disk cavitator</td>
<td>(Re_N=90\times10^6), (Fr_N=\infty), (C_Q=0.5)</td>
<td>Cavity shape, cavitator drag</td>
</tr>
<tr>
<td>AC2</td>
<td>Artificially-Cavitating Cone Cavitator</td>
<td>(Re_N=3.2\times10^7), (Fr_N=72), (C_Q=1.0)</td>
<td>Cavitation loads</td>
</tr>
<tr>
<td>AC3</td>
<td>Artificially-Cavitating Body</td>
<td>(Re_N=56,000), (Fr_N=26.7)</td>
<td>Cavity-body interactions</td>
</tr>
<tr>
<td>VC1</td>
<td>Twisted Hydrofoil</td>
<td>(Re_c=1.146\times10^6), (\sigma_\infty=1.07) &amp; (\sigma_\infty=\infty)</td>
<td>Complex cavity shapes, loads</td>
</tr>
<tr>
<td>VC2</td>
<td>Cryogenic Hydrofoil Cavitation</td>
<td>(Re_c=9.1\times10^6), (T_\infty=83.06K), (\sigma_\infty=1.7)</td>
<td>Cavitation with thermal sensitivities</td>
</tr>
<tr>
<td>VC3</td>
<td>Oscillating, Cavitating Wing</td>
<td>(Re_c=5.0\times10^5), (\sigma_\infty=0.6) &amp; (\sigma_\infty=4.0)</td>
<td>Dynamic &amp; load validation</td>
</tr>
<tr>
<td>FS1</td>
<td>Submerged Hydrofoil</td>
<td>(Re_c=1.45\times10^4), (Fr_c=0.57), (\alpha=5^\circ)</td>
<td>Submerged Body</td>
</tr>
<tr>
<td>FS2</td>
<td>Surface-Piercing Hydrofoil</td>
<td>(Fr_c=0.55), (Re_c=2.26\times10^6)</td>
<td>Surface-piercing body</td>
</tr>
</tbody>
</table>
Chapter 4

Accuracy Assessments and Validation Efforts

The baseline numerical procedure described in Chapter 2 is assessed by its application to supercavitating and free-surface flows. The objective is to better understand the strengths and weaknesses of the numerical scheme and evaluate its ability to predict metrics of engineering utility. Initial studies of the baseline numerics are performed, which includes the numerical-flux scheme and developing an improved understanding of the preconditioning algorithm. Next, appropriate practices for the computational meshes are developed and examined with regard to the flows of interest. Validation studies using physical experiments are also performed, which demonstrate an ability to predict the flows observed experimentally. Confidence in the ability to predict these flows and insight into future algorithm development are gained through these studies.

4.1 Assessment of Numerical Schemes

The numerical scheme has a large impact on the predicted interface, and thus, upon the corresponding flow field as well. For this reason, the numerical schemes are examined to give insight on the best available numerical approach.
4.1.1 Numerical Scheme Evaluation

Various forms of the numerical flux are investigated in their application to the artificially ventilated, disk cavitator test case (Case AC1). As the spatial accuracy relates to assumptions made in these numerical flux formulations, a consistent study can be useful in determining the better scheme for this particular application. The examined numerical fluxes include the Roe flux-difference-splitting and two AUSM-style flux-splitting schemes, AUSM+up and LDFSS. These approaches are reviewed in Chapters 1 and 2. Comparisons of the predicted volume-fraction profiles as a function of the radial distance, at two axial stations, are displayed in Fig. 4-1. Note that in this case a physically occurring interface is expected to be infinitesimal and that the reentrant behavior should occur far downstream in this infinite $Fr_N$ case.

Figure 4-1: Predicted volume-fraction profiles through a cavity interface using various numerical schemes. The comparison displays the interface-sharpness preserving ability. This is for the axis-symmetric-disk cavitator (Case AC1) at $Re_N=90\times10^6$, $Fr_N=\infty$, $C_Q=0.5$. 
First, consider the effect of the spatial accuracy of the Roe scheme. The 2\textsuperscript{nd} and 3\textsuperscript{rd} order accurate schemes compare well with each other, at least in their ability to maintain a sharp interface. Both resolve the interface within 3 to 7 cells, which is consistent with results for gas dynamics. With first-order-accurate interpolations, as expected, the predicted interface becomes more diffuse. Clearly, a higher-order scheme is required based on this study.

Next, 3\textsuperscript{rd}-order-accurate predictions using the Roe, AUSM\textsuperscript{+}-up, and LDFSS schemes are compared to both of the AUSM schemes. A less diffuse interface is predicted using the Roe scheme. For fluid-interface capturing, the AUSM schemes appear to behave somewhere between the Roe 1\textsuperscript{st}- and 2\textsuperscript{nd}-order-accurate schemes. Note that both of the implemented AUSM-based schemes do not encapsulate all features suggested in Edwards \textit{et al.} \cite{Edwards} or Liou \cite{Liou}, nor are they fully calibrated; such modifications are required prior to making any general conclusions. In any case, this smearing of the interface is undesirable and is physically interpreted as artificial mixing at the interface. Such mixing dampens large-scale structures and incorrectly enhances the air entrainment from the cavity. Artificially introducing this behavior is not desired. Other schemes, such as the inter-gamma differencing scheme \cite{81}, show a similar interface sharpness preserving capability \cite{37}, albeit, with a more stringent Courant number limitation of 0.3.

A more crucial measure of the prediction quality is the cavity profile, which is the dominant feature that influences vehicle and appendage loads. Solutions of the CFD cavities for the axisymmetric disk cavitator configuration (Case AC1) are compared to the semi-empirical cavity predictions (described in Eq. \textit{1.18}) in Fig. \textit{4-2}(a-b). It is clear
that with the 3rd-order Roe-scheme the predicted profile agrees well with the semi-empirical solution; whereas both AUSM schemes and the 1st-order Roe schemes are in poor agreement. In this comparison, although the ventilated gas rates are constant, the predicted cavity pressure, i.e. $\sigma_c$, differs amongst the solutions. For this reason, the $\sigma_c$ value from the 3rd-order accurate Roe solution is used to obtain the semi-empirical cavity solutions. To remove the ambiguity of a prediction that favors the Roe-based prediction, a comparison with prediction-specific $\sigma_c$ values are used for the theoretical cavity profiles in the comparisons in Fig. 4-2(c). The predicted error in the cavity profile with the case-specific $\sigma_c$ values actually worsens for the AUSM and low order schemes. Using the 3rd-order accurate Roe approach the predictions maintain an interface that is both sharp and has a cavity profile that is in good agreement with semi-empirical theories. For these reasons, this scheme is used throughout the remaining investigations.
In a dual-time integration approach, artificially introduced pseudo-time derivatives must approach relatively small values in order to recover the solution to the underlying governing equations. When these artificial terms are large, artificial sources and sinks are introduced into the governing equations. The first developed formulations use a steady preconditioning approach [62]; the method that is used within this work. In general, the ideal formulation for steady and unsteady problems differ. When a steady
preconditioner is used for unsteady problems, several hundred pseudo-time iterations are required for full convergence in pseudo time. This issue has been addressed by Potsdam et al. [63] using hybrid steady/unsteady preconditioning approaches. Regardless of this known deficiency, a steady preconditioner is used and further examined here.

For steady problems, with the use of a steady-preconditioning approach, the conservation of mass of each species is only guaranteed when the solution is converged in pseudo time. As this may require hundreds of pseudo-time iterations, this may not be a practical as the computational time could become prohibitively large. Although many of the investigated flows are unsteady, the unsteady regions are typically confined to low-momentum air regions that suggest that the unsteady effects are negligible. Here, the sensitivity of the species mass loss and cavity predictions are examined with respect to the integration effort in pseudo time.

The axisymmetric disk cavitator case (Case AC1) is used as a test to the sensitivities to the integration method in time. A comparison of the computed results to the semi-empirical solutions is displayed in Fig. 4-3. The cavity profile predictions, error from the theoretical cavity, and the volume of ventilated air lost through the cavity length are all plotted for unsteady and steady integration methods. In the unsteady cases, the number of inner-cycle iterations in pseudo time is varied. The integration method, steady or unsteady, does not affect the predicted cavity profile as apparent in Fig. 4-3 (a). However, as highlighted in Fig. 4-3 (b), the conserved mass is sensitive to the integration technique and effort. The steady integration appears to have good mass conservation properties; however, the unsteady cases show loss of the air mass at the fore part of the cavity that diminishes with increased inner cycles. It is expected that an unsteady, or
hybrid unsteady/steady, preconditioner would alleviate this issue for unsteady flows. Other studies of confined flows show a similar behavior, but exhibit a stronger dependence on the number of nonlinear integration steps (See Appendix B.1).

At this point, it is useful to consider the impact of, and strategies to eliminate, the described issues in the efficiency of the integration in time. The flows of interest are, in general, high-speed flows where the resident time of the air particles is relatively short. Thus, mass conservation from the slow convergence in pseudo time displays little impact on the predicted cavity shape. Many of the features of cavitating and free surface flows are relatively steady. Thus, the steady preconditioner remains sufficient and the incurred errors are small. Issues remain in the unsteady regions, for example at breaking waves,

Figure 4-3: Comparison of the semi-empirical and CFD cavity-profile predictions using various temporal integration methods. Part (a) is the predicted cavity profile, given by the $\alpha=0.5$ isosurface. In part (b), the volume of gas lost is plotted. This is for the axisymmetric-disk cavitator (Case AC1) at $Re_N=90\times10^6$, $Fr_N=\infty$, $C_Q=0.5$. Note that the axial distance in the x-axis is implied for all figures.

At this point, it is useful to consider the impact of, and strategies to eliminate, the described issues in the efficiency of the integration in time. The flows of interest are, in general, high-speed flows where the resident time of the air particles is relatively short. Thus, mass conservation from the slow convergence in pseudo time displays little impact on the predicted cavity shape. Many of the features of cavitating and free surface flows are relatively steady. Thus, the steady preconditioner remains sufficient and the incurred errors are small. Issues remain in the unsteady regions, for example at breaking waves,
reentrant jets, or dynamic-body simulations, where it is possible that approach becomes less effective. This study, and the one presented Appendix B.1, suggest that the present method is not appropriate for all unsteady flows, for example in a tank sloshing flow, where the resident time of the particles is infinite and maintaining mass conservation is crucial. Note that this conclusion is based only on fact that the computational expense is high to conserve mass, and that another approach may be more efficient. Recall that these issues have been resolved in Potsdam et al. [63]. The methodology of Potsdam et al. [63] relies on the more flexible, flux-vector splitting (i.e. AUSM) approach, which, as shown in the previous section, tends to dissipate the interface more than the flux-difference splitting scheme. It seems plausible that with their more flexible framework the AUSM schemes can be tuned or designed for interface sharpness preserving. Or perhaps, another preconditioning approach such as proposed all-Mach number Roe schemes [82] may provide a better overall scheme.

4.2 Grid Studies

It is crucial to understand the appropriate grid-resolution requirements for the class of problems of interest for a specific numerical scheme. Without such an understanding, grid generation is based on traditional rules, rather than case-specific rules. In general, the mesh requirements for multiphase and cavitating flows are quite different from other applications; thus, using a single-phase mesh can lead to poorly resolved flow fields and improperly interpreted results. Investigations of the mesh resolution for artificial cavitation, natural cavitation, and free-surface flows are
performed to improve the understanding of the requirements for a Roe-based shock-capturing scheme used to capture fluid interfaces.

### 4.2.1 Grid Study Series 1: Artificial Cavitation

*Cavity-Shape Prediction: Supercavitating Disk Cavitator*

Spatial grid studies are examined on the same notional, axisymmetric disk cavitator considered earlier (Case AC1). Again, comparisons to well-established, semi-empirical solutions are used to validate the simulations. Force predictions for various grids are displayed Table 4-1 below. Note that the drag predictions require corrections to achieve the drag at $\sigma_c=0$, and this is performed using the linearized disk-cavitator drag behavior versus cavity pressure, or $c_D = c_{D,0} + \frac{d c_D}{d \sigma_c} \sigma_c$ [13, 4]. Table 4-1 indicates that the CFD drag predictions are near the representative experimental drag value [13]. This experimental value is reported with an error that is based on the variability in the various experiments. The CFD predictions are within five percent of the experimental measurements for any of the meshes examined. Note that these correspond to the cell-size in the radial direction that is maintained through the cavity, and that typical cell sizes used throughout the remaining studies are on the order of $\Delta v/R_N$ of 0.02, which based on this study appear to be well within the asymptotic convergence range as the error oscillates around -0.5%.
The cavity shape predictions also show excellent agreement with semi-empirical theory. For this cavitator configuration, comparisons and errors in the predicted cavity profile as functions of the axial distance are displayed in Fig. 4-4. Fig. 4-4 (a) compares the predicted cavity radius to the semi-empirical prediction at various mesh resolutions. Note that the semi-empirical relations use the cavity pressure from the fine-grid solution. Fig. 4-4 (b) displays the error from the semi-empirical solution, whereas Fig. 4-4 (c) displays the error from the semi-empirical solution using the grid-specific $\sigma$ value. Near the cavitator, all solutions show good agreement with the semi-empirical cavity. Further downstream, increased deviations occur with the decreased grid resolution, but all are in reasonable agreement. Although an effect of mesh resolution exists, it is weaker than the observed effects from the spatial accuracy and numerical scheme.

Table 4-1: Force prediction with respect to spatial resolution for a disk cavitator.

<table>
<thead>
<tr>
<th>$\Delta x/R_N$</th>
<th>$C_D(\sigma=0)$</th>
<th>$C_D$ Error (%)</th>
<th>$\sigma$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2x Coarse</td>
<td>0.16</td>
<td>0.858</td>
<td>4.64%</td>
</tr>
<tr>
<td>Coarse</td>
<td>0.08</td>
<td>0.819</td>
<td>-0.08%</td>
</tr>
<tr>
<td>Medium</td>
<td>0.04</td>
<td>0.816</td>
<td>-0.46%</td>
</tr>
<tr>
<td>Fine</td>
<td>0.02</td>
<td>0.818</td>
<td>-0.29%</td>
</tr>
<tr>
<td>Exp.</td>
<td>-</td>
<td>0.82</td>
<td>3.0%</td>
</tr>
</tbody>
</table>
4.2.2 Grid Study Series 2: Natural Cavitation

*Naturally, Cavitating Twisted Wing Profile*

The grid practices required for a naturally cavitating flow are expected to vary from ventilated cavity simulations. Cavity shapes formed via natural cavitation correlate to the resolved pressure field, which differs from ventilated cavities where air entrainment is crucial. Refinements for the twisted wing using an NACA 0009 cross section (Case VC1) are localized to the wing-conforming O-grid that maintains a near-
isotropic mesh in the cavity-dynamics-resolving regions. As successive coarsening and refinement is not performed, the grid study is not fully consistent and is not expected to display monotonic convergence in all features and parameters. A summary of the various O-grids are provided in Table 4-2. Note that a consistent water-tunnel-resolving mesh is used through these studies.

Table 4-2: Description of various O-grids used for the hydrofoil.

<table>
<thead>
<tr>
<th>Grid</th>
<th>Foi perimeter Cells</th>
<th>Chord-normal Cells</th>
<th>Span-wise Cells</th>
<th>Isotropic Cell Size in Cavity</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>108</td>
<td>23</td>
<td>19</td>
<td>0.04c</td>
<td>Halved from Medium O-grid extends 1.0c $\Delta s_L=0.002c$ $\Delta s_T=0.004c$</td>
</tr>
<tr>
<td>Medium</td>
<td>215</td>
<td>45</td>
<td>37</td>
<td>0.02c</td>
<td>Shown in Fig. 3-5 O-grid extends 1.0c $\Delta s_L=0.001c$ $\Delta s_T=0.002c$</td>
</tr>
<tr>
<td>Fine</td>
<td>215</td>
<td>64</td>
<td>21</td>
<td>0.01c</td>
<td>Halved from 2x Fine O-grid extends 0.5c $\Delta s_L=0.002c$ $\Delta s_T=0.004c$</td>
</tr>
<tr>
<td>2x Fine</td>
<td>439</td>
<td>127</td>
<td>39</td>
<td>0.005c</td>
<td>Less refined spanwise O-grid extends 0.5c $\Delta s_L=0.001c$ $\Delta s_T=0.002c$</td>
</tr>
</tbody>
</table>

Table 4-3 displays the predicted loads compared to those of the experiment for the noncavitating predictions ($\sigma_v=\infty$) [83]. The single-phase solutions deviate from measurements in lift by about four to seven percent. This is reasonable considering the experimental uncertainties and the assumptions made through these simulations. This number remains consistent over large variations in the mesh and shows little mesh dependence.
Now consider the results from similar grid studies on a cavitating case, at $\sigma_v=1.07$, which is provided in the lower subsection of Table 4-3. In this study, a $k-\varepsilon$ DES turbulence model is used. Both cases are unsteady simulations where the averaging is performed over 20 or more convective lengths, at similar time-step sizes, after the lift history shows relatively recurrent responses. Comparing the predictions as the meshes are refined, the general trend in the predicted lift in Table 4-3 appears to be converging towards the experimental measurements. The best prediction, however, remains 12% lower than measured. Note that errors associated with various assumptions remain, i.e. thermodynamics models, boundary-layer characteristics, and water-tunnel-wall boundary layers. In general, the unsteady cavitating solutions show relatively strong sensitivities to the mesh resolution, hence, a need for a large number of computational cells are generally required for these cavitating flows. In this case, the solutions are only approaching the asymptotic range at the finest mesh. The shedding frequencies, displayed as a Strouhal number based on the lift and chord length, show to have converged to a rather consistent value for the finest two meshes in these studies.
The effects of grid resolution from this study are generalized in the cavity-shape predictions displayed in Fig. 4-5. These shapes are based on time averages of the unsteady solutions. As these are DES simulations, it is expected that the large-scale turbulent structures converge, which is in fact observed. The predicted cavity shape appears to be highly dependent on the mesh resolution in the cavity regions. Thus, improved mesh resolution tends to improve the resolved cavity dynamics and the general solution quality. The size of the predicted cavity tends to be a strong function of the leading-edge resolution. Even with high cavity-region resolutions, inadequate leading-edge resolution produces too small of cavity. Monotonic force convergence is not apparent (recall Table 4-3); this is a result of a nonsystematic refinement. In general, the cavity shape appears to be converging with additional grid resolution, suggesting that improved solutions would be obtained with additional refinement.

Figure 4-5: Effect of mesh resolution to the time-averaged flow field and cavity shape. The rear-view predictions are represented in the upper figures, whereas the view of the suction surface of the wing is given in the lower figures. The cavity is displayed using the blue and grey translucent isosurfaces at vapor-volume fractions of 0.5 and 0.1, respectively. The wing geometry is colored by $c_{p,b}$, and the streamlines are colored in red, which pass over the suction surface, and grey, which pass under the pressure surface. This comparison is for $\sigma=1.07$ (Case VC1) and using a DES approach.
The predicted time-averaged pressure distributions along the centerline are displayed in Fig. 4-6, which yields some insight as to why the lift is underpredicted. With grid refinement, the pressure approaches that measured in the experiments. In general, the fine-grid trend is in reasonable agreement with the measured values. Note that reported experimental pressures contain several nonrealistic mean pressures in the cavity region. The pressures are reported to be significantly less than vapor pressure and even below absolute zero; such unrealistic pressures are omitted. It appears that the pressure surface values are insensitive to mesh refinement. On the suction surface, the predicted pressures at the cavity location, ranging from the suction peak to the midchord, are slightly higher than measured. In this region, a reentrant jet occurs and mesh refinement tends to improve the predictions. This further supports the argument that the predictions are strongly influenced by the prediction of the cavity dynamics.

Figure 4-6: Comparison of the predicted, time-averaged, pressure distributions at the midspan of the twisted hydrofoil using various mesh resolutions. This comparison is for $\sigma=1.07$ (Case VC1) and using a DES approach.
4.2.3 Grid Study Series 3: Free-Surface Waves

The mesh requirements for subsurface objects are examined for a submerged NACA 0012 hydrofoil (Case FS1). Here, refinement is segregated into two studies, in the free-surface normal and axial directions. Successive halving in these directions are used to determine the importance of capturing various features. Sensitivities to refinement in the free-surface normal direction, that is on the blue mesh in Fig. 3-10, indicates an importance in maintaining interface sharpness. Alternatively, sensitivities to axial resolution imply that resolving the wave-induced flow beneath the free surface is crucial, that is axial refinement in the black mesh of Fig. 3-10. In these studies, all other meshes are held consistent.

In Fig. 4-7, the mesh level in each case is normalized by the coarse-mesh cell size for comparison purposes, i.e. $\Delta x/\Delta x_{\text{Coarse}}$. Therefore, in both cases the coarse-mesh resolution is indicated by 1.0. For the free-surface normal refinement case, $\Delta x_{\text{Coarse}}=0.05c$ in the free-surface normal direction at the interface. In the axial resolution studies, 0.5 is represented by the mesh shown in Fig. 3-10. The predicted wave-damping rates appear to correlate with both the free surface and wave-induced flow-field resolutions, as shown in Fig. 4-7 (a). Note that the wave damping rates should be low as the nonbreaking water waves examined have a minimal amount of dissipation. The predicted wave-amplitude appears to be unaffected by the water-field prediction as presented in Fig. 4-7 (b), but the free-surface sharpness remains important. From Fig. 4-7 (c), it appears that the mean-wavelength predictions are dependent on both the ability to predict a sharp free surface and the wave-induced flow.
The impact of the wave resolution on the forces and moments on the hydrofoil are also displayed in Fig. 4-7 (d-f). No data is available, thus, convergence is examined towards the solution obtained with the finest computational mesh. In general, the lift tends to be a stronger function of the free-surface resolution. This makes sense in that the free surface modifies the hydrofoil-surface pressures from the various wave-induced pressures that are dominated by the first wave. The drag remains a function of both the local free-surface elevation and the wave-induced field, where the wave field effects are likely encapsulating the wave drag induced by the submerged body. In general, lift appears to be insensitive to the prediction of the waves away from the body, whereas the drag shows sensitivities to the predicted wave train formed from the body.

Figure 4-7: Sensitivity of mesh resolution on wave predictions (a-c) and the loads on the submerged body (d-f). The blue ‘x’ indicates mesh refinement only in the free-surface normal direction, whereas the black ‘o’ represent mesh refinement only in the axial direction.
The results of these studies reveal where there is potential room for improvement in the numerical scheme. In this nonbreaking wave case, improvement associated with the free-surface resolution could be handled through interface sharpening. Methods such as level set, VOF, and adaptive mesh refinement are all suitable approaches. Another important point is that a well-resolved water field beneath the free surface is essential in simulating the wave dynamics. This is displayed as normalized stream-wise pressure gradients shown in Fig. 4-8. Note that the differential hydrostatic pressures due to the waves are as high as 10 percent of the free-stream dynamic pressure. The numerical dissipation of these wave-induced currents can also alter the wave dynamics. In any discretization method, the accurate prediction of the free surface waves relies on the resolution of the underlying wave-induced effects. In terms of adaptive-mesh refinement (AMR), both Hay and Visonneau [37] and Rhee et al. [43] note difficulty in achieving fine-grid-solution quality using an AMR that isolates the adaptation to the interface. These wave-induced motions may give insight into deficiencies of previous AMR algorithms, and suggest that a more accurate approach would incorporate subsurface motions into the refinement criteria.
4.2.4 Grid Study Series 4: Multiphase Turbulence Simulations

Mesh resolution dependencies for turbulence simulations are investigated for the naturally cavitating, twisted wing geometry (Case VC1). Recall, from Table 4-2, that the cell size in the cavity and large-scale-turbulent regions are varied. The predicted and theoretical (for isotropic turbulence) decay of the turbulent pressure spectra are plotted in Fig. 4-9 at the various grid resolutions using a consistent time-step size. These probes are positioned within the reentrant region of the cavity, away from the symmetry plane. As expected, there is a monotonic convergence and extension of the predicted inertial range with increased grid resolution. Using finer meshes, the theoretical decay is approached. The deviations observed from the theoretical turbulence behavior is not necessarily unexpected, as multiphase flows do not normally exhibit isotropic turbulence; regardless, the deviation is relatively small. This anisotropic behavior, occurring as a jump in the spectra near $fc/U=8$, is likely a result of the different energy levels associated with the

Figure 4-8: Stream-wise, non-dimensional, pressure gradient ($c/q_{\infty} dp/dx$) due to developed wave train.
different density fluids. The high-energy behavior occurs at lower frequencies, which correlate to turbulent water structures. The high-frequency, low-energy scales associated with the smaller-scale turbulent structures occur in the water-vapor mixtures. These results are reassuring as they indicate the expected decay behavior of the turbulent scales, and the proper convergence with mesh refinement gives further confidence in the methods. It is promising in the validity of the approach for cavitating-fluid flows.

![Figure 4-9: Pressure spectra at various grid resolutions. The pink line is the theoretical value for isotropic turbulence, \((fc/U)^{5/3}\). With mesh refinement, the simulation of additional inertial-range turbulent eddies are apparent and the behavior converges to the theoretical behavior.](image)

4.3 Time Resolution Requirements

The time-step size required for the examined flows is an important aspect. For a stable twin-vortex cavity (recall the diagram from Fig. 1-3), the cases are often run steady
state. This assumption is reasonable considering that the unsteady regions of the flow only occur in the low-momentum air, whereas the high-momentum water is rather steady. The time-step size required for a marginally stable cavity, where hysteresis is present in the $C_Q$-$\sigma_c$ curve, is examined using a three-dimensional disk cavitator at a $C_Q=0.3$. The effect of time-step size is displayed in Table 4-4, which shows little error in the cavity pressure when the time step size is less than 0.1 $U_\infty/D$. For greater values, the cavity tends to prematurely destabilize and this leads to large errors. If the time-step study is performed in nonhysteretic region of the $C_Q$-$\sigma_c$ curve, time-step size has little or no effect. In the hysteretic regions of the $C_Q$-$\sigma_c$ curve, the solution integration technique is crucial. Where, if the temporal integration is based on steady assumptions or uses too large of a time-step size, the solution always tends to the smaller cavity. When applying steady integration techniques, particularly if the solution sought is for a specified $C_Q$, rather than $\sigma_c$, it is crucial to understand these unsteady integration requirements.

<table>
<thead>
<tr>
<th>$\Delta t$ $(U/D)$</th>
<th>$\Sigma$</th>
<th>Error Est (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.05</td>
<td>0.0286</td>
<td>0%</td>
</tr>
<tr>
<td>0.1</td>
<td>0.0284</td>
<td>-1%</td>
</tr>
<tr>
<td>0.4</td>
<td>0.183</td>
<td>540%</td>
</tr>
<tr>
<td>1.6</td>
<td>0.185</td>
<td>547%</td>
</tr>
</tbody>
</table>

**4.4 Validation of Methods**

These studies of the numerical behavior carried out above provide some validation of the computational methods. Experimental verification comparisons are also
needed to highlight issues that do not arise in the numerical verification. These are especially important to validate the underlying models and assumptions.

4.4.1 Artificially Cavitating Flow Validation

The first validation set is based on artificially cavitating fluid flows. As a reminder, these cases are characterized by ventilating a noncondensable gas into the cavity. The reader is reminded that summaries of these cases are provided in Table 3-1 and a description is provided in Section 3.2.

*Load and Cavity Predictions: Cone Cavitator, Case AC2*

The ability to predict the loads on a supercavitating vehicle is a crucial aspect of the code validation and is a prerequisite for its application to design. Here, the 15-degree cone-cavitator (Case AC2, additional details are provided in Table 3-1 and Section 3.2.2) is used to validate the cavitator-load prediction ability. This case is representative of a cavitator on a full-scale, supercavitating vehicle. Such varied configurations include the loads necessary to encapsulate the required lift for controlling and trimming a vehicle.

Validation is performed over a range of angles of attack and computed lift and drag are compared to experimentally [71] based correlations [13]. The correlation accounts for a variable cavitation number, removing the need to iterate until a specific cavitation number is achieved. The cavitation number for these cases is roughly 0.03. The predicted loads are compared to the experimental correlation in Fig. 4-10. Note the excellent agreement in the predicted lift and drag with respect to the experiments. The
predicted cavity shape at 0 degrees, with $\sigma_c=0.08$ and using the coarse mesh, compares well with the only available experimental photo, at $\sigma_c=0.1$, in Fig. 4-10 (a) and (b).

![Image](image.jpg)

(a) Exp. [71]: $\alpha=0^\circ$, $\sigma_c=0.10$
(b) Predicted: $\alpha=0^\circ$, $\sigma_c=0.08$

(b) Comparison to experimental correlation.

Figure 4-10: Compared of CFD predictions to experiment for a supercavity forming around a 15°-half-angle conical cavitator (Case AC2). The observed cavity (a) is quite similar to that predicted (b) at a similar $\sigma_c$ value. The predicted cavitator loads (c) compare well with correlations to measurements developed by May [13].

*Artificially-Supercavitating Body Prediction, Case AC3*

In Case AC3, the computed supercavities with on-body closure effects are examined (details are provided in Table 3-1 and Section 3.2.3). The predicted cavity shape exhibits good agreement with experimental results, as can be observed in Fig. 4-11. Specifically, the effects of the strut are predicted quite well, including the stagnation region of the strut pushing the cavity away and upwards onto the vehicle body. Furthermore, a reentrant jet is induced on the opposing side of the vehicle from the strut,
which is also captured in the CFD simulations. Note that the experimental cavitator and body experiences small vibrations due to the strut [72] that could easily cause the angled feature of the reentrant jet apparent in the photograph. As no vibrations are modeled, the numerically predicted reentrant jet aligns perfectly in the reverse direction of the flow. Predictions of the $C_Q\sigma_c$ behavior are also in good agreement with the reported values in Schauer [72]. This level of agreement relies heavily on proper turbulence modeling; this is discussed in more detail in Chapter 6. These predictions demonstrate an ability to accurate simulate body-cavity interactions.

![Figure 4-11: Comparison of the cavity predictions to observed cavities from experiments [72] at similar cavitation numbers. The CFD-prediction geometry is colored by the pressure, and the blue, black, and grey isosurfaces are at vapor-volume fractions of 0.97, 0.8, and 0.05, respectively. The photographs are borrowed from Schauer [72].](image)

### 4.4.2 Naturally Cavitating Flow Validation

The next validation set is based on naturally-cavitating fluid flows. This is the type where phase change occurs from pressure decreases. Summaries of these cases can be referenced in Table 3-1 and details are in Section 3.3.
Load and Cavity Prediction: Twisted Wing, Case VC1

The load prediction capability on the previously discussed twisted foil case (Case VC2) is first considered without cavitation. The reader is referenced to Table 3-1 and Section 3.3.1 for additional details in the experiments.

Noncavitating: $\sigma_v=\infty$

Comparisons of the predicted forces are compared to the experimental measurements are presented in Table 4-5. The lift predictions are within seven percent of those measured. This is well within the uncertainty levels in this experiment. Specific uncertainties can be observed in the data. For example, a 5% increase in $C_L$ to from $Re_c=0.8\times10^6$ to $1.146\times10^6$ is observed when the wing is not cavitating and pitched at negative two degrees. In this Reynolds number range, lift is normally sensitive to the transitional boundary-layer behavior and similar sensitivities should be observed in a transitional XFOIL analysis. The XFOIL predictions (presented in Appendix B.6) display that lift is insensitive to Reynolds number for this cross section. This indicates the minimal amount of uncertainty in the experiments. Furthermore, the predicted mean lift curve slope ($\Delta C_L/\Delta \alpha$, in radians) using two CFD predictions is $2.23\pi$. This value compares reasonably with the theoretical value of $2\pi$. However, the local experimental lift-curve slope is $2.57\pi$, where over the mean over the entire range examined in the experiments is reported as $2.18\pi$. As the foil is not separating, such a nonlinear deviation in the lift-curve slope is not expected to occur. This suggests an additional local experimental error, or possibly wall effects, that can be contributing to such discrepancies. Regardless, for this case, the CFD predictions are in reasonable agreement with the measured values.
Although not measured, the CFD drag predictions are expected to be in error due to poor boundary-layer assumptions.

Cavitating: $\sigma_c=1.07$

Consider the same twisted-wing geometry, pitched at negative two degrees, but cavitating. In this case, regarding the cavitation model, the mass-transfer timescale is assumed to relate to the free-stream velocity and the hydrofoil thickness ($t_\infty=0.09c/V_\infty$). Past experience suggests that this value yields reasonable results for hydrofoils; however, some cases may display moderate sensitivities to this value. The water vapor is modeled with a density of 0.1 kg/m$^3$, roughly 4 times higher than actual. This still simulates a 10,000:1 density ratio. In many investigations, the water liquid-to-vapor density ratio is modeled at 1000:1; while maintaining accurate predictions compared to experiments [8,56]. At this point, the sensitivity to the density ratio is expected to be minimal on the cavity dynamics. Solutions for the medium-grid solution, using DES and an isothermal compressible model are provided in Figs. 4-12 and 4-13, where the predicted cavity shape is compared to experimental photographs for $\sigma_c=1.07$. Note that the predictions and experiment are not synchronized in time, but they are in the cavity location and size. Time synchronization is difficult to achieve because the large-scale shedding structures

Table 4-5: Predicted loads on the twisted wing compared to the measured lift.

<table>
<thead>
<tr>
<th>$\alpha$ Base (deg)</th>
<th>$C_{L,P}$</th>
<th>$C_{D,P}$</th>
<th>$C_{M,P}$</th>
<th>$C_L$</th>
<th>$C_D$</th>
<th>$C_M$</th>
<th>$C_{L,EXP}$</th>
<th>$C_{L,EXP,\text{ORR}}$</th>
<th>Percent Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>-1</td>
<td>0.561</td>
<td>0.0075</td>
<td>-0.148</td>
<td>0.560</td>
<td>0.015</td>
<td>-0.147</td>
<td>0.609</td>
<td>0.597</td>
<td>6.16%</td>
</tr>
<tr>
<td>-2</td>
<td>0.439</td>
<td>0.0067</td>
<td>-0.116</td>
<td>0.439</td>
<td>0.014</td>
<td>-0.112</td>
<td>0.468</td>
<td>0.459</td>
<td>4.42%</td>
</tr>
</tbody>
</table>
are chaotic in nature. In general, the predicted solutions contain shedding cavity structures quite similar to those in the experiment.

Fig. 4-12 contains views of the suction surface of the numerical predictions alongside photographs from the experiments. Both the experimentally observed and computed cavities exhibit a more stable region towards the leading edge and, to either side of the centerline, tend to extend further in the chord-wise direction. The centerline contains an unsteady region where a periodically shedding reentrant jet occurs. As a result of mesh coarseness relative to the actual scales, the main structure of this shedding event is more apparent in the CFD prediction. Approaching the trailing edge, the shed structure evolves into a hairpin vortex, which is a feature also observed in the experiments. In comparing the CFD cavity structure to the actual cavity, one evident deficiency is the lack of cavitation along the leading-edge suction peak towards the outer span of the wing. Low-pressure regions outline the experimental cavity, but the predicted cavity does not extend to those locations. The capture of this feature is addressed with improved models of the thermodynamic behavior in Chapter 6.

In Fig. 4-13, an isometric view of the predicted cavity is shown along with a corresponding comparison of the side view to the experimental photographs. Note the good qualitative agreement with the experimental cavity shape at various stages in the cavity-shedding cycle. The predicted cavity appears to be the same height as observed experimentally. The capture of the hairpin vortical structure formation, along with its location and advection are also well predicted.
Figure 4-12: Sequence of CFD solutions (left), from view of the suction surface, through nondimensional time ($t^* = t U/c$) using isothermal-compressible DES assumptions for the twisted wing at $\sigma_v = 1.07$ (Case VC1). The blue and grey translucent isosurface are at vapor-volume fractions of 0.5 and 0.1, respectively. At the right are photographs borrowed from experiments of Foeth et al. [73].
Figure 4-13: Sequence of CFD solutions, from orthogonal (left) and side (center) views, through nondimensional time ($t^* = tU/c$) using isothermal-compressible DES assumptions for the twisted wing at $\sigma_v = 1.07$ (Case VC1). The blue and grey translucent isosurface are at vapor-volume fractions of 0.5 and 0.1, respectively. At the right are photographs borrowed from experiments of Foeth et al. [73].
The time-averaged suction-surface-pressure predictions are shown in Fig. 4-14 (a). Note the increased pressures along the centerline, relative to just outside the centerline. This is an effect of the unsteady reentrant jet impingement and the unsteady cavity feature previously observed. Comparisons of the measured and predicted pressures on the surface are shown in Fig. 4-14 (b). Several of the reported experimental pressures are far below the vapor pressure and even fall below zero absolute pressure. These values must be nonphysical. In large-scale cavitation, in the presence of presumably adequate nucleation sites, the time-averaged pressure is physically limited to values above the saturation pressure; that is with the exception of small-scale time and space events. In no way could a subzero static pressure be measured.

These pressure measurements indicate that an error exists in the pressure acquisition process, or even more likely, in the reported $c_p$ values. One possibility is to assume that the wrong free-stream velocity used in the normalization of pressure to compute the $c_p$ values. Thus, the data are corrected such that a value of 5.74 m/s is used in Fig. 4-14 (c). Considering the test matrix of these experiments use velocities in the range of 5 to 6 m/s, such an explanation is far more likely than the measure of the impossible, subzero absolute pressures reported. With this correction, a better comparison is apparent. This includes the increased pressure along the centerline due to the jet-impingement event and a gradually decreased pressure moving towards the outboard regions. This uncertainty in the measure of pressure in the experiments makes it difficult to determine the location/features that are causing the simulation to produce a lack of lift as presented in Table 4-3. Improved lift predictions may require additional mesh refinements and fewer physical assumptions.
Figure 4-14: Predicted pressure distributions. Pressure contours on the suction surface are given in part (a). Comparisons of the CFD predicted pressure to data reported [83] at various span-wise locations, are provided in part (b) and a corrected form in part (c). Note that the corrected form assumes an error in the velocity used for the reported, nonphysical, $c_p$ values.
The ability to predict thermal effects associated with vaporous cavitation is evaluated. This is important for cryogenic rocket propulsors as well as evaluating thermal effects associated phase change in naturally cavitating flows. The computed liquid nitrogen flow over a cavitating hydrofoil is compared with measurements from experiments by Hord [75]. The hydrofoil is pitched at $\alpha=0$, has no camber, uses a leading edge radius of 0.396 cm, and has a chord length of 6.35 cm. In this case, significant tunnel blockage effects occur. These cases are performed using a compressible simulation with a fully populated state equation and a $k-\varepsilon$ unsteady RANS model.

There are two factors that complicate this particular problem. First, the unsteady inflow velocity resulting from the strong tunnel interactions with the cavity and the need to specify pressure at the inlet makes the boundary conditions iterative. Secondly, the thermal sensitivities in the fluid properties and saturation curve create strong sensitivities that relate to adequately modeling the thermodynamics. The combination of these two features results in a highly nonlinear cavity response to the inflow conditions. The inlet conditions achieved in these simulations are close to the experimental conditions of $Re_c=8.6 \times 10^6$, $T_x=83.06K$, and $\sigma_x=1.772$. Those reported at the experimental inlet were $Re_c=9.1 \times 10^6$, $T_x=83.06K$, and $\sigma_x=1.7$. This disparity exists as a result of the computed solution locking in to these free-stream conditions over a significant range of total pressures. It is expected that either the two-dimensional or the geometric assumptions may have prevented the solution from achieving the exact experimental conditions. After
achieving an unsteady recurrent solution, the predictions were obtained using unsteady integration with a time-step size of 0.5 µs, and an overall integration time of 5.0 ms.

Fig. 4-15 contains time-averaged representations of the flow field over an integration time of 5.0 ms. Fig. 4-15 (a) displays a “representative” photograph from the experiments, but for a hydrogen flow at different conditions. Note that measurements exist for nitrogen, but such a qualitative comparison to photographs is only available for nitrogen. Although this comparison is of different conditions and fluids, it is still valuable as it reassures the low-vapor volume fractions in the predicted cavity are expected in when comparing to the “representative” photographs that display a nontranslucent (or bubbly) cavity. This smearing of the cavity interface is due to the local temperature decreases and the corresponding $p_v$ reduction, which is the cause for these bubbly cavities Fig. 4-15 (b). Note that the time averaging could also cause some of the smearing. Following this initial cavity, low values of vapor volume fraction are sustained downstream of the cavity. This sustained cavity is observed in the shadows that appear behind the cavity in the experimental photo. The time-averaged pressures and temperatures are also plotted. A temperature depression within the cavity and the subsequent temperature rise at the aft of the cavity closure point are observed.
Time-averaged measurements from the experiments are available for comparison, which were taken at various locations along the hydrofoil chord [75]. In Fig. 4-16, the experimental and computed pressure coefficient, temperature, vapor-volume fraction, and the standard deviations of the CFD results are plotted. In addition to the time-averaged values, the CFD extrema are also included. The predicted pressure profile remains in reasonable agreement. However, the mean CFD pressure recovery is more gradual than measured. The experimentally measured temperature gradually rises from a minimum in the cavity region to values near that of the free stream at locations downstream of the cavity. Unlike the experimental measurements, the CFD temperature profile is not smooth, which is likely a result of span-wise vortical structures predicted in the flow. It is expected that this behavior would be alleviated by allowing the span-wise vortices to evolve in a third dimension. Also, the simulations show a rise in the temperature at the
cavity termination point that was not apparent in the experiment. This could be an issue with the resolution of temperature measurements in the experiments, or another artifact introduced by two-dimensional assumptions. Regardless, the prediction is in reasonable agreement with the experiment. The visually reported cavity has a length of 1.9cm, where based on the $\alpha_v$ profile, these predictions yield a 2.0-2.5cm long cavity. This is considered to be appropriate considering that the visual cavity length is represented by an ambiguous volume-fraction value.

**Figure 4-16**: Time-averaged solutions of a cavitating hydrofoil in a cryogenic (liquid-nitrogen) flow (Case VC3), at $Re_c=9.1\times10^6$, $T_\infty=83.06K$, and $\sigma_c=1.7$. The surface profiles for the pressure coefficient, temperature, vapor-volume fraction, and the standard deviations (of those measures) are provided.
4.4.3 Free-Surface Flow Validation

The final validation set is based on free-surface flows. In these studies, interactions with submerged bodies, relevant to submerged supercavitating bodies, and free-surface breaking geometries, relevant to water-entry cavitating bodies are examined. Again, these cases are referenced in Table 3-1 and details of the setup and experiments are given in Section 3.4.

*Submerged Hydrofoil, Case FS1*

CFD predictions of the wave profiles induced by a submerged NACA 0012 are compared to experimental measurements in Fig. 4-17 [78]. Two of the cases reported by Duncan are examined, a nonbreaking- and a breaking-wave case. The intent is to validate the CFD method for similar problems, for example, submerged cavitating vehicles. These simulations use steady, $q$-$\omega$ RANS formulations. The mesh used is provided in Fig. 3-10. The CFD solutions using two forms of the inviscid, numerical flux are presented to evaluate the effect on the predicted waves. The nonbreaking wave profiles are displayed in Fig. 4-17 (a) for the predictions and those measured in the experiments. The predicted amplitudes using CFD are slightly lower than those measured in the experiments, and the predicted wavelength and overall trend are in fair agreement with the experiment. These wave-heights and the increased dissipation are due to the smearing of the interface and a lack of wave-field resolution. In part (b) of Fig. 4-17, results for a similar, but breaking-wave case are presented. The breaking wave is formed by decreasing the submergence depth from 1.29$c$ to 0.783$c$. These results use a grid with a similar density to the nonbreaking-wave case. These predictions also agree well with the experimental
measurements. In general, the present predictions display a similar agreement with the experiments as those using an interface-tracking method [15], the level-set method [51], or with the Gamma-based differencing scheme enhanced with adaptive mesh refinement [37]. This supports the use of the presented CFD capability for free-surface flows.

Recall that the numerical flux had a significant impact on the predicted cavity shape. Here, another examination of the numerical flux for free-surface flows is displayed; this is consistent with the studies from the artificial supercavitation case. When using the LDFSS flux [34] in this submerged hydrofoil case, a better prediction of the amplitudes of the initial waves is observed at the expense of an over-damped wave train. The damping with the LDFSS scheme is a result of the more dissipated free surface, a feature displayed in the earlier evaluations of the numerical flux in Section

Figure 4-17: Comparison of the experimental measurements [78] and the predicted free-surface-wave profiles formed by a submerged NACA 0012 hydrofoil (Case FS1) at $Re_c=1.45\times10^5$, $Fr_c=0.57$, and $\alpha=5^\circ$. In part (a), the foil is submerged 1.29c below the nominal free surface and creates a nonbreaking wave, and in part (b), a breaking wave is formed when the foil is submerged 0.783c below the nominal free surface.
4.1.1. These results support the usage of the methodology for submerged bodies interacting with a free surface.

*Surface Piercing, NACA 0024 Hydrofoil, Case FS2*

Validation of the surface-piercing hydrofoil case (Case FS2) should yield confidence in the ability to predict the flow around surface-piercing objects. The case is listed in the summary in Table 3-1 and described in Section 3.4.2. These are unsteady, \( k-\varepsilon \) DES simulations computed on the mesh in Fig. 3-12. The CFD and experimental free-surface deflection measurements are displayed in Fig. 4-18. The general predicted free-surface shape is in good agreement. The crest ahead of the foil appears to be well predicted but looks weaker upstream than measured experimentally. The location of the trough is also in excellent agreement with the experiment. The predicted trough deepens to values close to the experiment, but seems to be covered by the incoming wave. With better mesh resolution, the breaking wave features can be better resolved and the improved interface sharpness can be better maintained to improve the predictions.
In Fig. 4-11, the predicted waterline is compared to the experiment using both qualitative and quantitative measures. The prediction of the waterline is shown in Fig. 4-19 (a), where the lines marked with “x’s” represents the measured values. The steady waterline up to the trough compares well with the experiment; thereafter, the unsteady wave profile remains between the maximum and minimum measured in the experiment. Finally, the general behavior observed in the photograph taken from the experiment, in Fig. 4-19 (b), agrees with the predictions. It can be observed in the CFD prediction at the aft region of the hydrofoil in Fig. 4-18, the free surface appears elevate at the hydrofoil surface but is not apparent in the measurements. This attaching of the wave onto the foil surface appears to occur in experiments as well, which is observable in Fig. 4-19 (b).
In Fig. 4-20 compares the predicted and measured pressure distribution along the span of the wing. Note that in the experiments a finite wing was used, however, no wing tip is modeled in the CFD. This allows the minimum pressure peak from the experiment to decay towards the tip and cannot occur in the CFD where the wing joins into an inviscid wall. The predicted pressure contours are in good agreement with the experiments, that is, the locations and behavior of the pressure minima as well as the pressure recovery along the chord.
4.5 Summary of the Investigations

In this chapter, assessments of the numerical scheme were performed. These studies investigated the impacts of the numerical, inviscid flux scheme, and the preconditioning method on the predicted cavity shape. It was found that the cavity predictions are highly dependent on the numerical flux used, where the 3rd order accurate Roe scheme possesses a capability to accurately predict cavity shapes and long-range wave fields. The steady preconditioner resulted in a lack of mass conservation for unsteady problems. However, the mass loss did not affect the predicted cavity profile. Although improved preconditioning methodologies have been reported that could remove this deficiency [35,63,82], the current approach shows to not impact the interface for a
cavitating-fluid flow. It is possible that this is a result of the relatively short residence time of the fluids in the computational domain. However, for unsteady-confined flows where mass conservation is essential, improvements to the preconditioning approach may be required for efficient computations.

An examination of the mesh and time-step requirements was performed. In artificially cavitating flows, the mesh sensitivity was shown to be fairly weak. Alternatively, for a naturally cavitating flow the mesh requirements grow significantly. This appears to be associated with a need to resolve turbulent scales and pressure gradients near the inception point of the cavity. Also, free-surface mesh refinements were performed. Algorithm-specific refinements revealed the importance of retaining interface sharpness. The wave-induced water field also appeared important to the accurate prediction of long-range wave dynamics.

Furthermore, several open literature experiments that relate to the flows of interest were validated. In these cases, load predictions for an artificially ventilated cavitator show to be adequate for design purposes. Also, the cavity-shape predictions were in excellent agreement with experimental observations. In vaporous cavitation, the load predictions are expected to be accurately predicted with sufficient resolution, but are yet to be captured. Furthermore, cavitation in cryogenic fluids that are highly sensitive to temperature changes was validated using the CFD. This includes the ability to capture the cavity shape and the more bubbly cavities present in such configurations. Finally, the ability of the CFD approach to predict submerged and surface-piercing bodies interacting with a free surface was validated. These validation studies engender confidence in application of the CFD method over a broad range of flows.
Chapter 5

Numerical Method Enhancements for Developed Cavitation and Discrete Interface Simulations

The overall goal this thesis is to improve the fidelity of cavitating-fluid flow simulations and the simulations capability. These goals are achieved through improved mesh and numerical techniques. Mesh technologies are investigated to reduce the time required to generate a computational mesh, to improve the overall mesh quality, and to enable complex geometric and dynamic simulations. The mesh technologies investigated include overset and adaptively conforming grids. These methods are presented and applied to relevant applications, are shown to improve the simulation capability. Numerical-scheme improvements are used to increase the solution accuracy, thus, decrease the necessary number of cells for a high-fidelity simulation. This improvement is based on a reformulated level-set approach that is valid for compressible multiphase flows. These combined modifications improve the overall numerical scheme.

5.1 Overset-Mesh Approach for Multiphase Flows

In this work, the overset-mesh approach is implemented into UNCLE-M and is evaluated for the application to cavitating- and free-surface-multiphase flows. Previous multiphase solvers have incorporated the overset-grid approach using a single-phase level-set approach [18] and an interface-tracking algorithm [17]. In the present work, an interface-resolving scheme valid for fully compressible fluid flows with an arbitrary
number of species is introduced. The overset method removes mesh-boundary matching constraints thereby enabling added flexibility in the mesh generation. The benefits of the overset approach include: improved mesh quality; simplified parametric configuration studies; and a more flexible dynamic-mesh capability. These are only advantages if the method does not compromise the solution quality of the problems of interest. Since several of the cases presented in Chapter 4 use overset meshes, it has been established that the method returns accurate predictions of cavitating and free-surface flows. However, more details of the implementation, the impact to solution quality, and caveats of the method are now considered. These analyses lead to a better understanding of the appropriate overset-mesh practices as well interpolation strategies.

5.1.1 Description of Implementation

The implementation of an overset-mesh capability into UNCLE-M uses interpolation libraries, via DiRTlib [65], and domain connectivity tools, using SUGGAR [66]. SUGGAR is an external code that produces domain connectivity information, this domain connectivity information (DCI) includes information needed to establish the coupling a system of overlapping computational domains. DiRTlib is a solver-included library that uses the DCI information and applies the interpolations within the solver to couple the overlapping domains. In order to discuss details of the implementation into UNCLE-M, it is useful to first clarify some of the overset terms. Consider the following terms with respect to the business-jet example in Fig. 5-1.
Donor cells are cells that are overlapped and included in the interpolation stencil of the fringe cells in an overlapped computational domain. For example, in Fig. 5-1, the cells of the green mesh that are near the overlapped cells with the red dots of the cyan mesh are donor cells.

Field cells are ordinary cells in the fluid domain where the baseline numerical approach is applied; these cells are not directly affected by the overset-grid approach. In Fig. 5-1, these are the shown cells that are outside of the overlap regions.

Fringe cells are intergrid boundary points that link the solution between overlapping meshes. The number of fringe cells required is equivalent to the extent of the stencil.

Figure 5-1: Sample overset mesh for a business jet. The black nodes, or fringe points, are located at the domain boundaries of the green mesh and interpolate values from the cyan mesh. The red nodes, or fringe points, make an intergrid domain boundary for the cyan mesh and interpolate values from the green mesh. The cyan mesh has “out” cells within, and in the proximity of, the jet that are not shown. This figure borrowed from the PEGASUS website [84].
of the numerical scheme. These fringe cells overlap into another computational
domain and receive information through interpolations from the donor cells of the
domains it overlaps with. Their incorporation into the linear system formation is
performed by providing a Dirichlet-like boundary condition based on these
interpolations. These are highlighted in Fig. 5-1 using the red and black spheres that
are located on the nodes.

• *Out cells* are “out” of the computational domain. There are two cases when this
occurs: first, when a solid body overlaps the volume of a another computational cell,
partly or completely; second, when too much unnecessary overlap exists between
grids, this excessive overlap is often minimized by marking cells as out. In all cases,
these out cells are excluded from the flow-solution update procedure. These cells are
not pictured in Fig. 5-1, but would consist of the cells in the cyan mesh that are
excluded because they are physically inside of the jet.

Within UNCLE-M’s framework, most of the implementation effort is in
accommodating the linear system to accept cross-domain coupling and the existence of
“out” cells. A formal description of the overset approach being incorporated into the
computational equations, based on the linear system formed in Eq. 2.33, is applied as

\[
\left[ A^{-1} + L_{\text{RHS}} \left( I - A^{-1} \right) \right] A\Delta Q = -L_{\text{RHS}} R^m, \tag{5.1}
\]

where the cell-specific selection matrix, \( L_{\text{RHS}} \), has been introduced for the mixture
continuity, momentum, energy, and species-mass conservations as
The values of \( f_o \) depend on the overset-cell type and can be expressed as

\[
L_{\text{RHS}} = \begin{bmatrix}
  f_o & 0 & 0 & 0 \\
  0 & f_0 \delta_{ij} & 0 & 0 \\
  0 & 0 & f_o & 0 \\
  0 & 0 & 0 & f_0 \delta_{ij}
\end{bmatrix}.
\]

\[5.2\]

The values of \( f_o \) depend on the overset-cell type and can be expressed as

\[
f_o = \begin{cases}
0 & \text{for out cells} \\
0 & \text{for fringe cells} \\
1 & \text{for field cells}
\end{cases}
\]

\[5.3\]

As presented, the left-hand side of Eq. 5.1 requires matrix inversion for each cell. In practice, this submatrix inversion is superfluous; rather, a single subroutine forces the matrix on the left-hand side to the identity matrix. The fringe and out cells are then effectively removed from the linear system. The values of the flow variables, \( Q \), are interpolated from the donor cells prior to the computing the right- and left-hand sides of the linear system. Within the linear solver, at each Gauss-Seidel sweep, the values of \( \Delta Q \), at each fringe cell, are interpolated from the donor cell values to maintain coupling within the linear system.

The step-function behavior in the volume fraction that occurs across an interface has special requirements. The first is the need to use a monotonic-interpolation method to prevent the artificial introduction of a new local minima or maxima at the fringe cells through the interpolation. This parallels needs for monotonic cell-face interpolations and TVD, a concept discussed earlier in Section 1.3.1.2. Monotonic interpolations are incorporated using clipping in DiRTlib, or one of the monotonic-interpolation methods available in SUGGAR. The clipping approach, within DiRTlib, is based on clipping, or limiting, the interpolated value based on the minimum and maximum values of the donor-
member cells. Further attempts to improve the interpolation quality is to smooth the volume-fraction field by applying a transformation given by

\[ \alpha' = \frac{1}{2} \left[ \frac{\tanh^{-1} \left( \left( 2\alpha - 1 \right) f_k \right)}{\tanh^{-1} ( f_k )} + 1 \right], \]

where the constant, \( f_k \), is used to maintain bounded values of \( \alpha' \). The value of \( f_k \) should be close to unity, and is arbitrarily chosen to be 0.999999. A comparison of the \( \alpha, \alpha' \), and their derivatives are given in Fig. 5-2. An analytic recovery of the volume fraction follows the overset-based interpolations. This more linear profile is expected to improve the numerical accuracy of the interpolations.

![Graph showing \( \alpha, \alpha', \frac{d\alpha}{dx}, \frac{d\alpha'}{dx} \) vs. \( x/\Delta x \)]

Figure 5-2: Example of behavior of the volume fraction, \( \alpha \), and the smoothed volume fraction, \( \alpha' \), in the proximity of an interface. Note that \( \alpha' \) approaches a more linear function near the interface.

5.1.2 Examination of Overset Application for Multiphase Flows

Within this research, it was observed that overlapping meshes can impact the predicted interfaces and resultant cavities. The impact of an overset-grid system is
illustrated for the previously examined axisymmetric disk cavitation example (Case AC1). From such studies, the proper application of the method is better understood and important conclusions are drawn for improved solution fidelity.

A large part of the errors in an overset grid system is attributed to mismatched cell sizes on overlapping boundaries. The effect is examined using a Cartesian-like overlapped mesh positioned on the interface, just after the water separates from the cavitation. Comparisons of predicted interfaces using different mesh resolutions for the overlapped grid are displayed in Fig. 5-3. Note that the base mesh, displayed in red, is the same fine mesh used earlier in the validation of Case AC1. The overlapping mesh shown in blue has cell sizes that are factors of 0.5, 1.0, and 2.0 of the baseline mesh. This study examines the effect of cell-size discontinuities in the interpolation region. The zebra-style contour levels of the volume fraction clearly display the location and thickness of the predicted cavity interface (black/white regions). In general, the solutions from the medium and fine mesh in Fig. 5-3 (a-b) are comparable to the baseline in Fig. 5-3 (d). This indicates that the interface sharpness is maintained through overlapping boundaries. In the solution of the coarsened overlapped mesh, provided in Fig. 5-3 (c), additional smearing occurs in the coarsened overset region that sustains through to the baseline mesh downstream of the overlapped mesh.
It is apparent that without an added interface modeling methodology, the coarsest cell size encountered affects the interface sharpness. Although this specific case does not illustrate a huge impact, other cases display a completely altered solution from this smearing. It is advised that when using overlapping meshes crossing an interface, the cell size should not grow larger than the size intended to resolve the interface. When the interface is artificially compressed, i.e. using the level set approach [26] or artificial compression terms [30], this concern should be alleviated. Using these approaches, the

Figure 5-3: Grids and cavity interface predictions (shown in black/white regions) from the overset-grid refinement studies on the axisymmetric disk cavitator at $Re_N=90\times10^6$, $Fr_N=\infty$, $C_Q=0.5$ (Case AC1). The baseline mesh is displayed in red, and the blue mesh is the overlapped mesh with various resolutions.
effect of cell-size discontinuities across the overlap regions can be recovered with such sharpening.

The predicted cavity shapes with varied overlapping mesh densities are presented in Fig. 5-4. The baseline prediction appears to be in better agreement with the theoretical solution compared to either of the overset-grid predictions. Even with a refined-overlapping mesh, a small amount of solution degradation occurs. Numerous factors could cause this behavior. The likely culprit is conservation errors across the overlapping regions. It is important to point out that for all the investigated refinement levels, the predicted cavities remain in reasonable agreement with the theory-based cavity throughout the cavity length.

Figure 5-4: Effect of overset-grid refinement on the predictions compared to semi-empirical theory for an axisymmetric disk cavitator at $Re_N=90\times10^6$, $Fr_N=\infty$, $C_Q=0.5$ (Case AC1).
Although the simplified, Cartesian-aligned overlap configuration maintains adequate results, application-oriented overset configurations must also be investigated. A cavity-aligned, stretched, overlapping mesh is presented in Fig. 5-5 (a). In this case, the mesh density in the direction normal to the cavity interface is similar to that of the baseline mesh (red mesh). It is clear that the interface in this cavity-aligned overlapped-mesh case displays a sharper interface than the Cartesian-overlapped case. In the cavity predictions, displayed in Fig. 5-5 (c), the cavity-aligned-mesh predictions deviate from the baseline predictions when using the inverse-distance-based weighting method from SUGGAR. The improved interpolation method based on the linear-least-squares fit, with clipping, yields an improved solution for the cavity-aligned overlapping-grid case. These results display strong solution sensitivities to the interpolation method.
These sensitivities motivate examinations of the interpolated solution when using various mesh and interpolation schemes. Figure 5-6 displays interpolated values of the volume-fraction field for the disk cavitator case. Note that these plots do not represent UNCLE-M CFD solutions; they represent the interpolated solution (white contours) from a nonoverset, baseline-donor solution (black contours). The idea here is to take the

Figure 5-5: Effect of overset-grid alignment and interpolation method on the predictions, compared to semi-empirical theory for an axisymmetric disk cavitator at $Re_N=90 \times 10^6$, $Fr_N=\infty$, $C_Q=0.5$ (Case AC1).
solution from the baseline mesh, pictured in Fig. 5-3 (d), and interpolate the solution onto the Cartesian-based overset mesh in Fig. 5-5 (b). Congruent solutions imply high-quality interpolations. In Fig. 5-6, the two interpolation methods are tested and both display that the interpolated solution is consistent with the “donor” baseline solution in this mesh configuration.

Figure 5-6: Interpolation quality using various interpolation methods on a Cartesian–overlapped mesh. The white contours indicate the interpolated values and the black contours indicate the donor values.
Using the skewed-overlapped mesh, a similar comparison to the prior is performed. In this case, the solution from the baseline mesh, in Fig. 5-3 (d), is interpolated onto the skewed-overset mesh pictured in Fig. 5-5 (a). The results are plotted for the two interpolation schemes in Fig. 5-7. Again, this only tests the interpolation and not the solver; the interpolated solution is given using the white contours, whereas the baseline solution is displayed using black contours. Although the linear-least-squared-based interpolations remain in good agreement with the donor solution, the inverse-distance-based interpolations are in rather poor agreement. This poor interpolation onto this skewed-overlapped grid with the inverse distance method indicates the cause for the poorly predicted cavities observed in Fig. 5-5 (c). This displays the weakness of the inverse distance method applied as an interpolation strategy within the overset approach.
5.1.3 Overset Mesh Strategies

A general strategy for building overset meshes for cavitating and free-surface flows is proposed that alleviates the previously described issues. First, care in choosing the interpolation method is required for overset meshes in such flows; this includes applying a high-quality monotonic interpolation schemes. Not only is it required for
algorithm stability, it also shows to improve the reliability of the predictions. The second recommendation is to avoid overlapping meshes across interfaces; this minimizes the possibilities of errors associated with mesh-resolution changes at the overlapping regions. The implication is that one should consider the location of interface with respect to the coarsened mesh regions to ensure that the interface does not dissipate prior to the regions of interest. Beyond these guidelines, the strategies differ for free surface and cavitating cases. These specific rules are discussed below.

5.1.3.1 Cavitation Meshing Strategies

The elimination of overlapping boundaries coinciding with a cavity interface improves the reliability of the predicted cavity shape and loads. A general, successful, grid scheme presented in Fig. 5-8 uses a contiguous grid over the body. An overlapping boundary exists in the wake, where the interface has already broken into a bubbly mixture. Although maintaining a contiguous mesh is not required, it tends to relieve issues that may occur in a complex simulation. For example, stretching the mesh away from a cavity-piercing control surface presents a case where the mesh can coarsen ahead of the control surface. This is just one, of the many possible, circumstances that can cause issues in a complex geometry. In this particular case, a complex, overset-based configuration predicts the experiments well, as displayed in Fig. 4-11.
5.1.3.2 Free-Surface Meshing Strategies

For meshing free-surface interfaces, a local refinement approach is introduced that relies on the overset-grid method. Specifically, at the free surface a nonconforming Cartesian-like mesh is refined in a region whose thickness is determined a priori to contain the free-surface waves that will arise. An example of the basic strategy is shown for the submerged hydrofoil case in Fig. 5-9. Here the mesh has a coarse air-field resolution, a well-resolved free surface, and the rather high resolution of the water field. Cases can arise where the interface must intersect the overlapping boundaries, for example, for free-surface-piercing bodies. Recall the earlier examined surface-piercing strut, in Section 4.4.3, where the predictions remain in reasonable agreement with the experimental measurements. The general strategy yields results that are consistent with experimental measurements of the free-surface elevations and pressure fields on the foil surface.
5.1.4 Validation of Overset-Mesh Implementation into UNCLE-M

The validation work from Chapter 4 exercises the overset-mesh approach, the implementation, and demonstrates an adequate simulation capability for engineering purposes. Additional validation of moving body applications is still needed and is further assessed here. This includes dynamic single- and multi-phase applications. In these studies, the overset-grid approach handles the mesh coupling for moving-body simulations. Examples of a wing oscillating in a water tunnel and a hydrofoil operating
beneath a free surface are presented as validation of the dynamic-multiphase capability with overset grids.

5.1.4.1 Dynamic-Mesh Validation: Oscillating Wing in a Water Tunnel

Noncavitating simulations of an oscillating wing experiment (VC3) have been performed to verify the dynamic-overset-mesh simulation capability. The experiment is described in Section 3.3.3. This particular hydrofoil shape exhibits a laminar separation, with an unsteady, turbulent reattachment that dominates the hydrodynamics at the higher-end of the lift curve. This is a notoriously difficult separation type for CFD to simulate due to transition modeling requirements [85, 86]. An evaluation of the associated effects, errors, and corrections are provided in Appendix B.5. There, it is argued that errors in the predictions are associated with the transition-sensitive attributes of this hydrofoil and the CFD predictions are corrected to account for such behavior. Agreement with the experimental measurements is quite good, as is shown in Fig. 5-10. Note that the two-dimensional case is extended to three-dimensional loads via Prandtl’s lifting-line theory and is inline with the three-dimensional prediction. Also, the minimum predicted lift is higher in the CFD as a result of unsteady features introduced from increased pitch frequency.
5.1.5 Dynamic Validation: Oscillating Hydrofoil Submerged Beneath a Free Surface

Validative experiments for multiphase, dynamic-body simulations in the applications of interest are difficult to find. For this reason, validation is performed using the submerged, NACA 0012 hydrofoil case examined in the earlier validation of free-surface flows (Case FS1) [78]. Conveniently, the oscillating airfoil experiments of Lee and Gerontakos [87] use an identical geometry, and similar Mach and Reynolds numbers as the submerged hydrofoil. Although the physical setup is different, the results will only deviate slightly provided that the interactions with the free surface are relatively weak. In this case, the mesh from the submerged NACA 0012 hydrofoil simulations is used with an oscillating hydrofoil, and the mesh can be referenced in Fig. 5-9.

In Fig. 5-11, the predicted loads at various submergence depths of the oscillating hydrofoils at $Fr_c=0.57$ are compared to the experiments [87]. The simulated pitch cycle undergoes angles described by $\alpha=10^\circ+15^\circ\sin(2kt^*)$. Where $k$ is the reduced frequency and is defined as $\alpha \omega/2V$. The nondimensional time, $t^*$, is given as $Vt/c$. A single-phase
CFD simulation of the oscillating airfoil experiments [87] was performed; however, these predictions are omitted as they only vary slightly from the case submerged at a depth of \( d/c = 1.29 \). This indicates that the free surface has a small effect on the loads at that depth. Note that the buoyant force on the submerged hydrofoil is removed for this comparison to the airfoil experiments. In general, the \( d/c = 1.29 \) case is in reasonable agreement with the experiment. In fact, considering the assumptions of an unmodeled transitional behavior, altered surrounding geometry (upper tunnel versus a free-surface), and the lack of using an appropriate turbulence modeling approach for separated flow regions, the predictions are in good agreement with the experiments for dynamic stall simulation.

Figure 5-11: Comparison of experimental loads [87] from dynamic stall case compared to simulations of a submerged, oscillating hydrofoil at various depths.

### 5.1.6 Summary of Overset Approach Implementation

The overset approach displays the capability to extend to cavitating-fluid flow simulations. As in all strategies, care has to be taking in the developed mesh and
numerical approaches used. Essentially, accurate interpolations and care in the overlap regions intersecting with interfaces show to be crucial. Finally, the overset-mesh approach displayed an ability to simulate complex-dynamic configurations. Such a capability is important for modeling vehicle dynamics for supercavitating vehicles prior to experimentation.

5.2 Mesh Adaptation for Free Surfaces

An adaptive-mesh method that improves interface-capturing techniques is devised. The method is based on observations from Kunz et al. [24], where interface-aligned cells display an improved interface preserving ability over cells skewed with respect to the interface. Those results can be verified based on observations in Fig. 5-5, where the cavity-aligned mesh has a less diffuse interface than the Cartesian aligned mesh. An approach consistent with this observation is a volume-based, surface-tracking method. This may be considered as a sort of semi-Lagrangian-Eulerian approach. Such a scheme can provide a sharp interface with fewer computational cells for smooth waves and the volume-fraction field can handle the high-frequency content.

A surface-tracking technique that evolves with an interface can be handled in numerous ways. More obvious methods would track interface-defining quantities (i.e. \( \alpha = 0.5 \)), a range of an interface-defining quantity (i.e. \( 0.05 < \alpha < 0.95 \)), or some filtered fit to the interface such as a Fourier- or wavelet-based representation would provide. Any of these tracking methods should give predictions with a significant improvement over those
from a stationary-mesh method. In terms of implementation, tracking an interface-defining quantity is logically simplest, thus, is investigated here for evaluation purposes.

5.2.1 Description of Method

A surface-tracking algorithm was developed that efficiently increases the mesh resolution on a free-surface-type interface, while not burdening the effort in the overset-related computations. The steps involved are as follows:

1) Initial grid definition: The user defines a stretched, initially Cartesian, mesh that has a refinement band of cells for interface sharpness, centered on the interface, and has a thickness greater than the anticipated height of the largest-amplitude breaking wave. A vertex index, $j_{FS}$, located at the center of this refinement band moves in the vertical direction such that remains fixed to the interface. An example refinement band and the positioning of $j_{FS}$ with respect to the free surface are displayed in Fig. 5-12.

2) Interface tracking: The capabilities needed to accommodate a free-surface tracking capability are described in the following steps.

a. Overset Efficiency: For efficiency within an overset-based solver, a static band of cells is defined at the boundaries of the adaptive mesh, marked as “static cells” in Fig. 5-12. For static-body grid systems, this feature decreases the computational time by eliminating the need to reevaluate the overset-grid connectivity information, i.e. update the interpolation stencil and weights, at each iteration. However, in a dynamic mesh, the “static cells” are not needed as the domain connectivity step is already required for each time-step and no penalty is incurred.
b. **Surface Tracking:** The definition of the interface is first needed, in this case, $\alpha=0.5$. It must then be detected and the mesh is adapted in the vertical direction such that the $j_{FS}$ index conforms to the interface. Cases arise that complicate this tracking, such as a breaking waves, where multiple interfaces may exist. The interface, or interfaces, are detected by looping through $j$ and determining the number of instances and locations where the $\alpha=0.5$ value occurs. The breaking waves are accommodated by positioning $j_{FS}$ at the mean interface height as illustrated in Fig. 5-12.

c. **Stabilized Tracking:** A stable tracking procedure is maintained by under relaxing the mesh displacement, i.e.

$$y^* = (f_r - 1)y^n + f_r y_{Interface}$$ \hspace{1cm} 5.5

In this example, the superscripts $n$, $^*$, and $Interface$ indicate the current mesh location, the under relaxed updated mesh location, and the mean interface location, respectively. A relaxation factor, $f_r$, of 0.25 was used in these investigations. Furthermore, a Laplacian mesh-smoothing procedure is applied to the displacements as

$$y^{n+1} = y^* - f_l \nabla^2 (y^* - y^n),$$ \hspace{1cm} 5.6

where the superscript $n+1$ indicates the updated mesh position. The factor $f_l$ controls the amount of smoothing used, and a value of 0.1 is used. This smoothing is useful to prevent oscillatory mesh behavior.

d. **Blended Displacement:** A smooth transition of the vertical spacing, from the displaced cells to “static cells” is desired. A blended displacement technique, where
the vertical displacements are varied based upon the index with respect to the static cell index, yields a smooth transition of the cell spacing.

e. Interpolation: Rather than considering conservative geometric law terms, an interpolation procedure of the previous solution, onto the updated mesh is performed. Such a procedure is quick and minimizes the errors incurred in the mesh motion. This is only performed in the regions near the free-surface interface, where the adaptation occurs.

Figure 5-12: Sample of the mesh adaptation to a free surface with breaking waves.

In general, this method is well suited for free-surface flows. It parallels the traditional surface tracking approach, but by allowing the volume-fraction field to track breaking waves the method is less limited method for complex free-surface features. It is an improvement over the baseline approach as the mesh is more efficiently utilized.
Some general rules for meshing using the approach still exist. First, when $\Delta x << \lambda$, where $\lambda$ is the local wavelength, a single grid index conforms well to the interface. This may not be the case when $\Delta x > \lambda$. Also, recall from Section 4.2.3 that the accurate prediction of wave dynamics still needs a high axial and transverse resolution with respect to the wavelength. Secondly, the refinement band should be sufficiently thick to resolve breaking wave features occurring on the interface. In general, this approach requires less knowledge of the final solution a priori.

5.2.2 Submerged Hydrofoil

The submerged NACA 0012 hydrofoil (Case FS1) is revisited using an adaptive mesh strategy, where the grid adapts to the free surface as displayed in Fig. 5-13. This is an overset grid system specified with gradual mesh coarsening through the depth of the domain. It can be observed that the mesh is well behaved after the adaptation and that the high-resolution band adequately tracks the free surface.
In turbulent simulations, there tends to be a limitation of interface sharpness with increased resolution. Based on the studies of the underlying numerical scheme in Section 4.1.1, the finite thickness of the simulated interface is expected to behave as $5\Delta y_I$, where $\Delta y_I$ is the cell spacing at the interface. In this case, $\Delta y_I$ is set to 0.001$c$, where the wave height is roughly 0.05$c$. However, as displayed in the detail of Fig. 5-14 (a), the interface now requires as many as 25 cells to resolve it. The interface thickness of 0.025$c$ is of the same order as the predicted turbulence length scale of 0.0125$c$. Although physically an interface is infinitesimal, this behavior is quite physical in the ensemble averaging of the modeled high-frequency waves in a RANS methodology. For these RANS calculation, $\Delta y_I$ should not be much smaller than the turbulent length scale unless the interfacial-shear layer is of interest.

Figure 5-13: Free-surface adaptation for a submerged NACA 0012 hydrofoil (Case FS1).
5.2.3 Surface Piercing Hydrofoil

The adaptive grid approach is also applied to a three-dimensional free-surface-piercing NACA 0024 hydrofoil geometry (Case FS2). For these studies, the mesh is simplified from the previous validation, where now a single O-grid and a lateral plane of symmetry are used. This modeled domain is also much smaller. The total cell count for the adaptive case is roughly $1.6 \times 10^5$ cells, and the cell spacing at the interface is $0.0015c$. By comparison, the earlier, non-adaptive case had an interfacial cell spacing of $0.0125c$, and uses roughly $3.0 \times 10^5$ cells for an equivalent domain size. Using the adaptive-mesh
methodology enables that half of the cells can provide a factor of eight increase in the interface resolution.

The final adapted mesh is shown in Fig. 5-15. Note that the chosen tracking method is well suited for this geometrically simple case. This particular free-surface flow contains a wide range of wave frequencies, from a smooth to a breaking free surface, and the adaptation approach remains adequate. The mesh conforms well to the waves with long wavelengths, for example the bow wave, subsequent trough, and shoulder wave. The adaptation to the high frequency waves can be observed in the attached wave near the hydrofoil trailing edge.

Figure 5-15: (a) Adapted grid to the free surface of the surface-piercing NACA 0024 hydrofoil (Case FS2). (b) The predicted free surface, shown by the isosurface at $\alpha=0.5$, aligns with the adapted mesh. The black gridlines indicate the adaptation on the surface of the hydrofoil. The red gridlines indicate a slice through the $x/c=0.95$ location.
A comparison of the predictions of the adaptive and nonadaptive methods with the experiments is provided in Fig. 5-16. The adaptive-case prediction displays a better-resolved near-body wave field with fewer cells. Away from the hydrofoil, the horizontal resolution is poor for the adaptive case, thus, the predicted trough in this region is wider than measured. Note that the adaptive method has little smearing in the smooth interface regions and continues to predict a foamy mixture in the shoulder wave. This is an accurate representation of the interface behavior as can be observed in experimental photographs, which is discussed later. Note that the free-surface elevation measured in such mixture regions is difficult for such a numerical comparison. This is a result of the ambiguity of the measurement with respect to the CFD isosurface, where it is possible that the $\alpha=0.5$ isosurface will not coincide with the measurement. For example, in a foamy mixture the free surface will physically occur at gas-volume fractions greater than 0.5 because of the added air content in the water. For this reason, it is can be assumed that interface may occur at the $\alpha=0.75$ value in the mixture regions, and in the smooth regions, remains at 0.5. However, in both cases, the isosurface at is $\alpha=0.5$ given. The shallow, relative to measured, foamy wave in the wake is likely due to the ambiguous definition of the interface in the mixture regions.
In Fig. 5-17, the nondimensional pressure field with the hydrostatic pressure removed, i.e. $c_{pdyn} = (p_\infty - p_s - \rho g z)/q_\infty$, is displayed on the hydrofoil surface and can be compared to the measurements. Note that with the adaptive grid case, the low-pressure region at the trailing edge of the hydrofoil is more pronounced than predicted using the nonadaptive case. In general, a better agreement with the measurements in that low-pressure region is achieved with the adaptive mesh. This improvement is indicative of an improved prediction of the foil-free-surface interactions.
Side views of the predicted solutions and experimental photographs are provided in Fig. 5-18. The CFD prediction is in good agreement, that is in the physical behavior of the wave field and the predicted water line remains bounded by the measurements. Note that with adaptation the surface dynamics are more pronounced than without, refer to Fig. 4-19 for the nonadaptive solutions.

Figure 5-17: Predicted pressure distribution, $c_{p,dyn}$, compared with the experiment of the NACA 0024 surface piercing strut (Case FS2). (a) Experiment measurements [79], compared to (b) baseline and (c) adaptive predictions.
In the rear-quarter view, shown in Fig. 5-19, the solution is compared to experimental photographs. Below the surface, near the hydrofoil trailing edge, a well-documented wave-induced separation region occurs [79], which is displayed in the streamlines in Fig. 5-19 (a). The streamlines passing though this separation region are colored by the local air volume fraction displaying the high air content, or bubbles, that persists in this separation region. In this prediction, the air actually entrains from the wake region of the foil and follows a path that leads it into the shoulder wave. A similar view in Fig. 5-19 (b) displays the predicted mixing level on the $\alpha=0.9$ isosurface; where the foamy regions are highlighted using a measure of interface diffusion, $1/|\nabla \alpha|$, which displays the locations of the predicted breaking-wave regions. If this is compared to the photograph in Fig. 5-19(c), the high values of $1/|\nabla \alpha|$ coincide well with the observable air/water mixture or the white regions on the free surface. It is more obvious in the shoulder wave regions, where slightly elevated values also occur in the wake of the shoulder wave and on the bow wave. In general, this behavior is in excellent agreement
with the photograph displayed in Fig. 5-19(c). Based on the agreement with the experimental photograph, it is assumed that the predicted smeared interface is more physical rather than an artifact of numerical dissipation.

Figure 5-19: Quarter view of a surface piercing NACA 0024 hydrofoil (a) streamlines hydrofoil surface are colored by volume fraction (b) $\alpha=0.9$ isosurface colored by $1/|\nabla\alpha|$ to display mixture regions (c) photograph from experiments [79].
5.2.4 Summary of the Approach

In these studies, the baseline adaptation procedure has been presented and assessed. Although this approach shows improvement for local free-surface refinement, several issues with the method can be improved. First, complicated input logic and an interactive usage are required. These features suggest that the approach may not be well suited for general applications, however, the benefits are likely well suited for certain specific applications. Second, iteration is still required for an adequate mesh, and a topology-evolving mesh adaptation type is more desirable. Third, it is quite possible to evolve to a poor quality mesh that contains large stretching ratios that can lead to solution inaccuracies. Finally, regarding surface-piercing geometries, this adaptation procedure is only well suited for interface-perpendicular geometries. For example, ship hulls would require a more complex adaptation method with horizontal displacements. This is not the case for supercavitation applications. Regardless of these issues and limitations, the implementation is fairly simple and the concept of “interface conforming” adaptation can still be evaluated.

5.3 A New Hybrid Volume-Fraction Based Level-Set Approach

A new methodology based on the level-set approach is formulated. The proposed level-set formulation is based on the traditional method for interfacial-fluid flows [22]. However, the use of a characteristic function provides an extended capability. These concepts extend the method previously proposed by Olsson and Kreiss [28] and Olsson et al. [29]. In the present work, a method with an analytical equivalence to the level-set
method of Sussman et al. \cite{22} is approached. The value of this formulation is an improved understanding of the level-set technique, an understanding of mass conservation issues is further developed, and analytic transformations between variables and procedures of interest are derived.

5.3.1 Comparison of the Level Set and Volume Conservation Approaches

Transport Equation Equivalence

First, consider the comparison of the physical behavior of a level-set and volume-based transport equation scheme. This is an analytic comparison to evaluate the two forms of integrating the interface in time. The reader is referred to Section 1.3.1.1 for background on the level-set method applied to interfacial-fluid flows. Start with a scalar transport equation of the signed-distance function, \( \phi \), as

\[
\frac{\partial \phi}{\partial t} + \frac{\partial u \cdot \phi}{\partial x_i} = 0. \tag{5.7}
\]

In this form, the incompressible assumption is not invoked as in the traditional level-set formulations \cite{22, 23, 26}. The traditional level-set transport equation is arrived at, from Eq. 5.7, using the chain rule and incompressible assumptions as

\[
\frac{\partial \phi}{\partial t} + \frac{\partial u \cdot \phi}{\partial x_i} = \frac{\partial \phi}{\partial t} + u_i \frac{\partial \phi}{\partial x_i} + \phi \left( \frac{\partial u}{\partial x_i} \right)^0 = \frac{\partial \phi}{\partial t} + \nabla \cdot \nabla \phi = \frac{D \phi}{Dr} = 0, \tag{5.8}
\]

where the last term is the implied notation for a material derivative. Next, a \textit{tanh} Heaviside-step function approximation is introduced and described as
Here, \( k \) is a constant that controls the smearing of the \( \tanh \) function to better approximate the traditional Heaviside function from Eq. 1.4. Note that for \( \varepsilon=1.5\Delta x \), the error between Eq. 1.4 and Eq. 5.9 is minimized at \( k=0.379 \). The overall difference in these Heaviside functions, when \( k \) is near this value, is negligible. The benefit of this variation is that \( \phi \) can be explicitly expressed in terms of the Heaviside function as

\[
\phi(H) = k\varepsilon \tanh^{-1}(2H-1).
\]

Eq. 5.10 is a transformation of the volume fraction function to the signed distance function and Eq. 5.9 is the inverse. The Maclaurin-series expansion of \( \tanh^{-1}(2H-1) \) is given as

\[
\tanh^{-1}(2H-1) = (2H - 1) + \frac{(2H - 1)^3}{3} + \frac{(2H - 1)^5}{5} + \cdots = \sum_{n=1}^{\infty} \frac{(2H - 1)^{2n-1}}{(2n-1)}.
\]

In this case, \( \phi \) is only important over a small range, i.e. over \([-\varepsilon, \varepsilon]\). Substituting the solution of Eq. 5.11 into Eq. 5.10 and the result into Eq. 5.8 gives

\[
\frac{D\phi}{Dt} = k\varepsilon \frac{D}{Dt} \left[ \sum_{n=1}^{\infty} \frac{(2H - 1)^{2n-1}}{(2n-1)} \right] = 0.
\]

The level-set transport equation is now expressed in terms of the volume-fraction function. After successive chain rule applications (see Appendix A.2) the higher-order derivatives are removed and Eq. 5.12 simplifies to

\[
k\varepsilon \sum_{n=1}^{\infty} \frac{1}{(2n-1)} \frac{D}{Dt} (2H - 1)^{2n-1} = 2k\varepsilon \sum_{n=1}^{\infty} (2H - 1)^{2n-2} \frac{DH}{Dt} = 0.
\]
The series is again reduced using the Maclaurin series of \((1-X^2)^{-1}\), which is given by

\[
\frac{1}{1-X^2} = \sum_{n=0}^{\infty} X^{2n} = \sum_{n=1}^{\infty} X^{2n} + 1 = X^2 \sum_{n=1}^{\infty} X^{2n-2} + 1. \tag{5.14}
\]

The series term in Eq. 5.13 can be replaced with a modified form of Eq. 5.14 when rearranged as

\[
\sum_{n=1}^{\infty} X^{2n-2} = X^{-2}\left(\frac{1}{1-X^2} - 1\right). \tag{5.15}
\]

An analytic expression of the signed-distance-function transport equation in terms of the volume fraction is expressed as

\[
2k\varepsilon(2H-1)^2\left(\frac{1}{1-(2H-1)^2} - 1\right)\frac{DH}{Dt} = \Gamma\frac{DH}{Dt} = 0, \tag{5.16}
\]

where \(\Gamma = \frac{2k\varepsilon}{(2H-1)^2}\left(\frac{1}{1-(2H-1)^2} - 1\right)\) after simplification. Finally, after dividing out the leading term, the incompressible form of the mass-conserving volume-fraction transport equation is recovered as

\[
\frac{DH}{Dt} = 0 = (0 + T.E.)\Gamma^{-1}. \tag{5.17}
\]

The \(T.E.\) term accounts for differences expected to occur in the truncation error; these are normally small, but are different between the numerical solutions from the two transport equations. Note that the two transport equations remain identical, even in the truncation error, away from the interface where \(H \to 0 or 1\). Finally, the result that the two transport equations are identical is no surprise; physically, all source-less transport scalar equations
are identical regardless of the transported variable. The only differences occur in the numerical error and realizability constraints.

With this analytic equivalence, it is useful to evaluate the level-set method within the context of the solver in question, UNCLE-M. Several relevant features are missing in the algorithm at this point, which include mass transfer modeling and compressible fluid assumptions.

**Compressible Flow Considerations**

The level-set method requires special additions for compressible-fluid flows [26]. In this Section, an evaluation of the method with respect to fluid compressibility is performed. Consider the species-conservation equation for a compressible fluid, in a chain-rule expanded form, which is expressed as

\[
\frac{\partial \rho_{H=1}}{\partial t} + \frac{\partial \rho_{H=1} u}{\partial x_i} = \phi \left( \frac{\partial H}{\partial t} + \frac{\partial u_i H}{\partial x_i} \right) + \rho \left( \frac{\partial \rho_{H=1}}{\partial t} + \frac{\partial \rho_{H=1} u}{\partial x_i} \right) = 0. \tag{5.18}
\]

Note that \(\rho_{H=1}\) implies the temperature- and pressure-dependent density of the fluid indicated by \(H=1\), and that density variations in any other fluids are encapsulated within the mixture density used in the mass-conservation equation. The variables \(H\) and \(\rho_{H=1}\) are respectively equivalent to \(\alpha\) and \(\tilde{\rho}\) from Eq. 2.1. Transforming Eq. 5.18 into a signed-distance-function transport equation valid for compressible fluids gives

\[
\frac{\partial \rho_{H=1}}{\partial t} + \frac{\partial \rho_{H=1} u}{\partial x_i} = \rho_{H=1}^{-1} \Gamma^{-1} \left( \frac{\partial \phi}{\partial t} + \frac{\partial u_i \phi}{\partial x_i} \right) + \rho \left( \frac{\partial \rho_{H=1}}{\partial t} + \frac{\partial \rho_{H=1} u}{\partial x_i} \right) = 0. \tag{5.19}
\]

This equation can be rearranged to yield a transport equation for a signed-distance function that is suitable for the case where the fluids are compressible, given by
The right hand side implies a source term that is required for species-mass conservation. The coefficient of the first term implies that the term is only significant near the interface, thus, for compressible problems it is possible that this term become important. Neglecting this term is likely the cause of oscillations present in the application of traditional level-set approaches to shock-tube problems, and creates the need for the ghost-fluid method to eliminate such behavior [26]. As a solution method to integrate Eq. 5.20 is not so straightforward, it becomes obvious that the species-mass conservation form is better suited for compressible-multiphase flows.

**Finite-Rate Chemistry Modeling**

Another benefit of the analytical transformation is for the application of a finite-rate-chemistry model within a traditional level-set approach. Recall, from Eq. 2.1, that the chemistry modeling is accounted for in a species conservation equation using

\[
\frac{\partial \rho_{H=1} H}{\partial t} + \frac{\partial \rho_{H=1} H u_i}{\partial x_i} = \dot{\omega}.
\]

After making incompressible assumptions, the transformation to a signed-distance-function transport equation using an equivalent finite-rate chemistry model can be incorporated as

\[
\frac{D \phi}{Dt} = \frac{\dot{\omega}}{\Gamma \rho_{H=1}}.
\]  

5.21

In practice, it may be questionable to use this approach. An interface must be initially defined if a signed-distance function is to have any meaning, which is not generally required in applications using these chemistry models. Furthermore, \(\Gamma^{-1}\) tends to force the
source term to zero away from the interface, thus, it may require a more complicated and nonphysical initialization strategy.

**Reinitialization Equivalence**

As a result of the reinitialization step, differences occur between the level-set method and the species, or volume, transport method. This occurs through source terms that remove the dissipation at the interface in the level set method. The source terms are better understood through a similar transformation process. The reinitialization procedure, or source term, of the level-set method \[ \text{[22]} \] is normally

\[
\frac{\partial \phi}{\partial \tau} = \text{sign}(\phi) \left(1 - |\nabla \phi|\right). \tag{5.22}
\]

Using the transformation from Eq. 5.10, and, as done by Sussman *et al.* \[ \text{[22]} \], assuming that \( \text{sign(}\phi)=2H-1 \), Eq. 5.22 becomes

\[
\frac{\partial k \epsilon \tanh^{-1}(2H-1)}{\partial \tau} = (2H - 1) \left[1 - |k \epsilon \tanh^{-1}(2H - 1)|\right]. \tag{5.23}
\]

Using a similar strategy as before, applying the series-solutions expansions, Eq. 5.23 becomes

\[
\Gamma \frac{\partial H}{\partial \tau} = (2H - 1) \left[1 - \Gamma |\nabla H|\right], \tag{5.24}
\]

which reduces to

\[
\frac{\partial H}{\partial \tau} = (2H - 1) \left(\Gamma^{-1} - |\nabla H|\right). \tag{5.25}
\]

Eq. 5.25 is the level-set sharpening source term, which is expressed in terms of the volume fraction. This transformation is verified in Fig. 5-20 using analytic descriptions of the gradients for a one-dimensional case. Here, the interface is positioned at \( \epsilon=0.0 \) and
the signed-distance function assumes a functional behavior of $\phi=0.8x$. The right hand side of Eq. 5.25 (volume fraction based) is compared to that of Eq. 5.22 divided by $\Gamma$ (Signed Distance Func.). The analytic representations, in Fig. 5-20 (a), display analytic equivalence of the reinitialization procedures. This verifies that the transformation of the reinitialization equation is correct.

Next, consider the discretized problem, where the gradient terms are computed using numerically formed derivatives. In Fig. 5-20 (b), the resolution is sufficient to discretize the interface, and the right-hand sides remain similar. In practice, the resolution is typically on the order of that displayed in Fig. 5-20 (c), or $\varepsilon=1.5\Delta x$. In this case, using this volume-fraction form, the residual actually displays the inverse behavior than desired and the reinitialization behavior smears the interface. This appears to be a caveat of this particular approach, which can be alleviated in two ways. In the previous cases, the numerical derivatives are formed using central differencing. As displayed in Fig. 5-20 (d), a one-sided-based differencing scheme eliminates this behavior. Here, the differencing is always weighted in the direction of the interface. Note the increase in residual, which implies a more aggressive reinitialization procedure. The second possible approach is to transform the volume fraction to a signed distance function and compute the gradients on this smoother function. Either approach should be effective.
With this alternate formulation, insight into aspects of the reinitialization behavior can be examined. For species conservation to be maintained, it is apparent that \( \frac{\partial H}{\partial \tau} = 0 \) must be zero, which is a task attempted in the algorithm of Sussman [25]. Within the sharpening procedure, this constraint is impractical for each cell. This is a result of the direct relation between the signed-distance function and volume fraction, and the volume fraction and species-mass conservation. Thus, the only case that can yield a condition where \( \frac{\partial H}{\partial \tau} = 0 \) is the case where the reinitialization procedure is eliminated. However, it
could be possible to satisfy the mass conserving constraint at the local level, within the vicinity of the neighboring cells.

### 5.3.2 Methods of Reinitializing the Interface

Several level-set implementation approaches are investigated within a volume-fraction-based framework. The three methodologies investigated are based on solving Eq. 5.22, solving Eq. 5.25, and a completely alternative approach to defining the sharpening source terms.

**Level-Set Version 1 (LS-1): Signed-Distance-Function Reinitialization**

In this formulation, Eq. 5.22 is solved at a stage after the volume fraction field is integrated in time. In general, this requires the transformation from the volume fraction to a signed-distance function. Note that the transformation works reasonably well in the proximity of the interface, the only relevant region in the method. The general steps within the time-step loop include:

1) Transform the volume fraction field to the signed-distance function using Eq. 5.10. Note that for a bounded distance function, a scaling of the volume fraction should be used, or $\phi = k\epsilon \tanh^{-1} f_k (2H - 1)$.

2) Solve Eq. 5.22. This is solved using a 4-stage, Runge-Kutta integration technique. The spatial discretization uses a central-upwind, van Leer limited scheme. Within the limiting, the first-order approximation is weighted towards the interface.

3) Retrieve the sharpened volume fraction using Eq. 5.9.

**Level-Set Version 2 (LS-2): Transformed Signed-Distance-Function Reinitialization**
Rather than solving for a signed-distance reinitialization, the transformed volume-fraction reinitialization equation, Eq. 5.25, is solved. Again, this occurs at a stage after the volume fraction field is integrated in time. In this case, no transformations are needed. The numerical technique used to solve Eq. 5.25, parallels that used to solve Eq. 5.22.

**Level-Set Version 3 (LS-3): Realizable- Scaled Sharpening**

For interfacial flow applications, the basic idea of the level set method is to redistribute the fluid-representing variable such that a sharp interface is maintained. The most general requirement of the level-set method is to preserve the location of the interface. Other constraints loosely imply that a constant thickness interface is maintained; this is solely for numerical stability purposes and in application, by virtue of only periodic reinitialization [23], is by no means a strict constraint of the method. Simplified methodologies can easily be constructed that conform to this nonaltering interface-location requirement. One proposed method may use a source term that applies a simple scaling of the volume fraction. For example, consider that

\[
\frac{\partial H}{\partial \tau} = \min \left( \max \left( \frac{1}{2} \left[ \frac{H - 0.5}{0.5 - \epsilon_2} + 1 \right] - H, -H \right), 1.0 - H \right),
\]

where \( \epsilon_2 \) is a small number that scales the volume fraction equally about the interface. The effect of Eq. 5.26 is better presented as a postprocessing step after integration in time. The modification to the postintegration volume-fraction field, at the \( n+1/2 \) level, is updated as

\[
H^{n+1} = \min \left( \max \left( \frac{1}{2} \left[ \frac{H^{n+1/2} - 0.5}{0.5 - \epsilon_2} + 1 \right], 0 \right), 1 \right),
\]
where increased $\epsilon_2$ values tend to scale the volume-fraction field away from the interface, and small portions outside of the realizable limits. Outside of these limits, the volume fraction is clipped and small amounts of mass are simply neglected. The result is an interface that is sharpened without altering the $\alpha=0.5$ level set. The effect is displayed in the example presented in Fig. 5-21. In the ideal case, the amount of mass loss is small when the baseline numerical scheme is sharp prior to the artificial sharpening. The method is not conservative, but the implementation and computational efforts are minimal. Lastly, as a volume fraction, rather than a fixed distance, defines the interface, this particular method artificially sharpens such that sub-grid scale mixtures are not necessarily removed.

Figure 5-21: Example of the sharpening procedure for the realizable scaled-sharpening approach.
Improved descriptions of the implementation of these level set methods within UNCLE-M are provided in Appendix A.3. Note that although this is presented for a fully coupled solver, the implementation into a segregated-solution solver is similar.

### 5.3.3 Validation of the Method

**Test Case 1: Ventilated, Axisymmetric Disk Cavitator**

It is important that the method be validated for practical applications. Recall the axisymmetric, ventilated disk cavitator test case (Case AC1). In Fig. 5-22, the radius versus the volume-fraction profiles are given for various numerical schemes on the medium-resolution mesh. At the upstream location, where the baseline method retains a tight interface, the level-set sharpening has little effect. However, further downstream, the baseline scheme exhibits a more diffuse interface and all the level-set approaches display an interface resolved within 3-4 cells.

![Figure 5-22](image-url)  
*(a) x/R_N = 1.42 (b) x/R_N = 16.7*

Figure 5-22: Effect of the level-set formulations on interface-sharpness preserving for an axisymmetric disk cavitator (Case AC1).
The predicted cavity profiles, with grid refinement, are of greater interest in this validation study. Although the method sharpens the interface, the accuracy of the predicted cavity remains important. The predicted cavity profiles, for various mesh refinement levels and using the LS-1 approach, are presented in Fig. 5-23. In this mesh variation study, a factor of 2 in each direction is used. A similar study using identical meshes and conditions is available for the baseline scheme in Fig. 4-4. A comparison of the two methods indicates that the level-set approach predicts a more accurate cavity with fewer computational cells. In Fig. 5-23, the predicted cavity shapes only change slightly with coarsening until the 2X coarsened level. Using the baseline method, the errors gradually increase with mesh coarsening. Furthermore, the level-set predictions on the coarse mesh are roughly the same quality as the medium-mesh prediction using the baseline method. This implies a factor of 2 reduction, in each direction, for a similar prediction capability.
Similar predictions are presented for the remaining level-set schemes, versions 2 and 3, which are plotted in Fig. 5.24. These results display a similar predictive capability as the previous case, with mesh refinement. Based on this similar behavior it is concluded that the variations in the examined sharpening procedures do not have a significant impact on the solution quality. Thus, for reasons of simplicity and flexibility, the realizable-scaled-volume-fraction approach seems preferable. Note that this conclusion is based on steady simulations. It is possible that an unsteady simulation would require a different strategy.

Figure 5.23: Effect of the mesh refinement using level-set formulation 1, applied to an axisymmetric disk cavitator (Case AC1).
Figure 5-24: Effect of the mesh refinement using level-set formulations 2 and 3, applied to an axisymmetric disk cavitator (Case AC1).
The schemes are further evaluated by examining the convergence rates of integral parameters. The convergence of the predictions with mesh refinement is examined for the various schemes and is provided in the plots presented in Fig. 5-25. Note that the actual rate of convergence is also indicated on the plot. One measure is the predicted cavity shape, which is plotted in Fig. 5-25 (a). Here, a root-mean square (RMS) of the difference in the predicted cavity shape for the specific mesh, from the cavity predicted using the fine mesh is used as a measure of error. In this plot, the level-set methods predict an error in the medium mesh similar to that of the fine mesh of the baseline solution technique. Furthermore, the level-set simulations tend to improve the overall order of accuracy in the cavity-profile prediction. Another, integral, measure of the flow field is the drag coefficient at \( \sigma_c = 0 \), which is estimated using linear assumptions of the drag response to cavity pressure for a disk cavitator [13]. In Fig. 5-25 (b), are the errors in the predicted drag compared to the finest-solution drag with decreased cell sizes. Although the accuracy of the cavity predictions improved using the level-set method, it appears to reduce the accuracy of the drag prediction. Regardless, the convergence for this highly nonlinear problem remains near a formal second-order convergence rate using a level-set approach. It is reasonable to assume that with complex configurations, with added mesh distortion, the convergence to better solutions will rely more heavily on a better cavity prediction.
Test Case 2: Shock Wave-Interface Impingement

A similar test to that proposed by Osher and Fedkiw [26] is used to test the robustness of the computational approach for compressible-fluid flows. The test case sends a high-speed shock wave moving at $M=11.38$ into a gas-water interface. The domain is 10 m long, from -5m to +5m where the gas-water interface is located at 0 with the gas at $x<0$ and the water for $x>0$. This case has four separate initialization regions: for $x<-4.04$, $\rho=8.27 \text{ kg/m}^3$, $p=1.0\times10^7$, $u=2950 \text{ m/s}$; for $-4.04<x<0.0$, $\rho=1 \text{ kg/m}^3$, $p=1.0\times10^5$, $u=0 \text{ m/s}$; for $0.0<x<4.62$, $\rho=1000 \text{ kg/m}^3$, $p=1.0\times10^5$, $u=0 \text{ m/s}$; and for $x>4.62$, $\rho=1004.1 \text{ kg/m}^3$, $p=1.0\times10^7$, $u=-6.38 \text{ m/s}$. The computational mesh uses 500 cells. The gas uses the ideal-gas law, whereas the liquid uses a temperature dependent, linearized, relation for water (the relation in Eq. 6.3). Note that vaporization is not modeled in this event and is compensated for by setting a minimum pressure limit at the vapor pressure.

Figure 5-25: Convergence of the estimated error in the (a) computed cavity profile and (b) the $C_{D,0}$ values with mesh refinement. The rate of convergence across the examined range is indicated.
The solutions at a time of 3 ms are compared in Fig. 5-26 for the (a) pressure, (b) gas-volume fraction, (c) velocity, and (d) density. Note that the gas-volume fraction is overlaid on the solution for reference. The baseline solutions are based on the Roe flux-difference splitting approach. Roe’s approach is one of the more acceptable approaches for modeling high-speed shocks and with adequate resolution, thus, there is little doubt in the accuracy of the predictions. The level-set predictions are based on modifications in the volume-fraction field from the baseline method. When using such a volume-fraction level-set formulation, the formulation is based on physically conserving the mass of each species. Thus, the ghost-fluid method [26] is not needed to prevent oscillations, and the level-set strategies can still sharpening the interface.
In general, the trends and values of the solutions are in agreement with each other and the solutions presented by Osher and Fedkiw [26]. The LS-3 predictions show little deviation from the baseline in any variable, the location of the interface is not altered, and it maintains interface with fewer cells, as displayed in Fig. 5-26 (b). Recall that the LS-3 scheme does not specify an interface thickness as in the LS-1 and LS-2 approaches. In

Figure 5-26: Test case displaying the solution at 3 ms after a Mach 11.38 shock wave strikes a gas-water interface.
Fig. 5-26 (b) the LS-1 and LS-2 interface predictions are shown to use fewer cells than the LS-3 and baseline predictions. The interface predicted with the LS-2 method shows a slight movement compared to the others, which is expected to be resolved with tuning. However, the solution only varies slightly from that of the baseline predictions. The more noticeable differences in the solutions occur when using the LS-1 formulation, which is most obvious in the pressure and velocity fields. These effects imply an effect of interface sharpening on the resultant solution. If interfacial dissipation is not present, such as when striking solids or using a film to prevent mixing, the LS-1 or LS-2 methods are more appropriate. However, in interactions between fluids this is not necessarily true, at least on the resolvable scales. In this case, the LS-3 formulation seems most appropriate.

5.3.4 Summary of the Developed Level-Set Approach

A new level-set approach was developed for cavitating-fluid flows. The approach is based on a volume-fraction transport equation and acts to sharpen the interface based on the signed-distance function. Other approaches of reinitialization that are better suited to turbulent interfaces are also proposed. The approach also displays the ability to simulate compressible-fluid flows without the use of ghost-fluid cells. Finally, the approach improves the accuracy of cavity-profile predictions which should yield a more efficient simulation capability.
5.4 Summary of Numerical Method Improvements

The reviewed efforts yield an overall improved simulation capability. First, the overset method was evaluated for application to supercavitating flows. The method appears to be suitable for extend the solution capability, improve the mesh for more complicated geometries, and enable efficient dynamic simulations. Next, a volume-based mesh-interface-tracking method was developed and evaluated. Although the usage is quite complex, the strategy shows a potential method to improve free-surface-type simulations. Finally, a volume-based level-set method was developed and evaluated; the method is more efficient and flexible, and is better suited for cavitating flows. The level-set methodology significantly improves cavity predictions as the cell-requirement for accurate cavity simulations are reduced. The developed level-set method is applicable to highly compressible flows without the need of internal ghost-fluid cells. In general, the method shows promise for free-surface, cavitating, and general interfacial-fluid flows.
Chapter 6
Improved Physical Models for Cavitation Modeling

The effectiveness of several physical models is evaluated for the numerical prediction of cavitating-fluid flows. Steady cavitation problems are, in general, predicted accurately using standard methods. On the other hand, partially cavitating flows tend to be more troublesome. Thus, they are further evaluated with respect to physical model improvements. The first physical feature examined is the approach to modeling turbulent flow. The focus is evaluating the impact of simulating, rather than modeling, eddies in the inertial-range of the turbulence spectrum for cavitating-flow simulations. Next, thermodynamic models of the fluid properties and their impact on cavitating flows are examined. In both studies, the modeling approach affects the predictions and methods are outlined that are useful to overcome modeling deficiencies.

6.1 Hybrid RANS/LES Turbulence Modeling for Cavitating Flows

In the past few decades, improved models of turbulent flow behavior have been developed to the point where they can efficiently handle a wide variety of flows. In traditional turbulence modeling approaches, based on solving the RANS equations, inertial and dissipative eddies are modeled and the low-frequency turbulence is simulated in the unsteady RANS (URANS) simulations. Using RANS approaches enables quicker convergence and is more efficient for high $Re$ simulations. As a result of assumptions in
their formulation, RANS models are not well suited for all flows, for example, separated
flows. However, numerous variants exist intended to address these problems. These
RANS approaches are the current standard for CFD. The extension to homogenous
multiphase simulations results in a form similar to the modeling approach used in single-
phase flows. However, the validity of such models is somewhat questionable near
interfaces and in boundary layers with bubbles.

The alternative approach is directly simulating the turbulent structures. A varied
amount of modeling may exist in this approach, where for LES only the dissipative range
of the turbulent spectrum (smaller than the Taylor scale) is modeled. In DNS, all of the
turbulent scales are simulated. A challenge for LES simulations is the modeling of the
correct subgrid-scale turbulent dissipation, which behaves like the artificial dissipation of
most numerical schemes. This implies the need for scheme-specific calibrations to the
models. This attribute suggests another approach, referred to as Monotonically Integrated
LES (MILES), which is investigated within this work. In MILES, the modeling of the
subgrid-scale turbulence is provided implicitly through the artificial dissipation provided
by a flux-limited numerical scheme. It can be argued that the model discrepancies in a
MILES model are similar to those incurred from a LES approach that is not calibrated for
a specific numerical scheme, application, and even grid system. Essentially, any factor
that affects the artificial dissipation of a numerical scheme implies additional calibration
is needed. When applying the MILES model, the implication is that the dissipation
behavior in the turbulence is not affecting the large-scale turbulent structures.

A LES modeling approach has flow-dependent strengths and weaknesses. LES is
acceptable for most nonwall-bounded flows where the largest eddies typically scale on
the geometry size. However, for high-Reynolds number turbulent boundary layers, the turbulent eddies are much smaller, and their proper spatial and temporal resolution, increases the computational effort to the point that renders LES approaches impractical for naval design in the foreseeable future. Without satisfying these mesh and time step requirements, features such as turbulent separation cannot be accurately captured with LES. However, an LES approach is better suited towards simulating separated flow dynamics. In short, one must be conscience of the important features of the flow prior to applying the LES modeling approach.

With recent gains in computational power, turbulence modeling has been redirected towards simulating larger turbulent scales while modeling the smaller ones. Applying LES with wall functions and hybrid RANS/LES turbulence modeling approaches are some of the more common approaches. As for the LES approach using wall functions, the sublayer and buffer regions of a boundary layer ($y^+<100-300$) are modeled, but the simulation of boundary-layer turbulence outside of this region is still required. This is a formidable task for design calculations. The hybrid RANS/LES approach applies a RANS model, a two-equation-based in this context, that uses the mesh size, with respect to the eddy size, to blend into a Smagorinski-type LES model. This method of hybridization is referred to as the Detached Eddy Simulation (DES) methodology [53]. The DES approach is in no way limited to two-equation turbulence models, as the original form investigated a one-equation approach [88], and numerous variants exit. This particular approach is evaluated for cavitating flows. The implementation into UNCLE-M is described in Eqns. 2.6-2.11. Recall that within the homogenous mixture assumption, the method is similar to single-phase approaches.
However, numerical issues associated with large density ratios make it difficult to use the suggested hybridized differencing schemes of Strelets [53]. In general, the strategy presented here shows an improved simulation capability for the examined flows.

6.1.1 Artificial Supercavitation

An investigation of the turbulence modeling approach applied to a body ventilated with a noncondensable gas, at a flow rate corresponding to the reentrant flow regime (Case AC3), is performed. The water-based $Re_N$ of the experiments is 56,000. It is assumed that the free-stream turbulence intensity is 0.02, the flow is incompressible, and the water tunnel walls are specified as inviscid walls. The mesh uses isotropic cells in the wake, whereas the body-conforming mesh cells have roughly a 3:1 azimuthal to axial spacings. The wake resolution is a “proper” DES-quality grid, whereas near the body the grid resolution is not completely isotropic. Regardless, the large-scale turbulent structures are apparent in the predicted cavity shapes.

Fig. 6-1 displays comparisons between the predicted cavitation numbers to those measured, as well as samples of the predicted cavity shape compared to those observed in the experiments [72]. These predictions use a variety of turbulence modeling approaches that include RANS, URANS, and DES methodologies [53]. Furthermore, studies of geometric approximations are also presented. All of the cases were initialized from the high-$C_Q$ (~2.0) RANS solution.
First, results when neglecting the support strut are reviewed. The predicted cavity pressures at various ventilation rates are displayed in Fig. 6-1 (a). The RANS simulations clearly predicts $\sigma_c$ values that are elevated over those from the experiments in the range of $0.1 < C_Q < 0.32$, which indicate that the predicted cavities are small relative to experiments in that range. Unsteady simulations are also performed using both the URANS and DES approaches. A consistent time-step size and mesh are used throughout these studies, where the nondimensional time-step size is $\Delta t/U_\infty D_N = 0.1$. This time-step size is identical to that required for accurate disk cavitator simulations in the validation presented in Section 4.3. As occurred in the twin-vortex cavity in the disk-cavitator validation case, with any coarser time-step size the prediction reverts to roughly the steady-RANS cavity prediction. The unsteady cases predict a more agreeable $C_Q - \sigma_c$ curve with respect to the experiments. In fact, the URANS prediction is almost in perfect agreement with the experiments.

Figure 6-1: Effect of turbulence model compared to data and photographs from experiments by Schauer [72].
agreement with the experiments, whereas the DES prediction has an even lower cavitation number than the experiments.

In Fig. 6-1 (b) is a comparison of the predicted cavities and experimental photographs at a condition where the predictions are sensitive to the turbulence model and the time-integration approaches. As expected, the RANS cavity is small and both of the unsteady cavities (URANS & DES) agree well with the experimental photograph to the extent measurable by a visual comparison. Note that the predicted DES and URANS cavities are still inconsistent.

*Effect of the Support Strut*

Now, results incorporating the support strut into the model, representative of the experimental setup, are reviewed. Based on the lateral symmetry results, i.e. neglecting the strut, a URANS approach appears to provide the most accurate cavity prediction. However, the predicted URANS cavity is distinctly different from the DES cavity on a DES-adequate mesh and time-step size. This indicates that other factors may be affecting the study. By modeling the elliptical support strut, a better evaluation may be performed. In the specific experiments examined [12], the cavity shape was sensitive to the profile of the support strut. If the shape of the strut affects the observed cavity, it is expected that neglecting it may have a strong impact on the computed results.

Included in Fig. 6-1 (a), are several results including the support strut in the simulation, which provides a better representation of the experimental configuration. The URANS results lie closer to those of the high-$\sigma_c$, steady RANS prediction. The DES results, with the modeled strut, show $\sigma_c$ predictions that are more closely aligned to the curve from the experiments. Not modeling the strut clearly affects the cavity prediction.
It is concluded that near $C_Q=0.3$, where the DES results (neglecting the strut) deviate from the experiments, the impact of the strut is felt by the cavity. Thus, based on results when modeling this more representative experimental geometry, the DES approach provides an improved prediction capability.

An understanding of the impact of the strut on the solution is useful for interpreting the results. Recall the flow features from Fig. 4-11. A reentrant jet on the opposite side of the strut forms and a rather high amount of air exists in the wake of the strut. These are features that tend to increase the air entrainment rate, thus, decrease the cavity size for a given ventilation rate. A similar trend was observed experimentally [12], suggesting that a more streamlined elliptical strut enables lower-$\sigma_c$ cavities at lower $C_Q$ values than when using a nonstreamlined cylindrical strut. The fact that the URANS predictions without the strut agrees with the experiments, and with the strut the cavity is too small, indicates that the URANS approach is effectively increasing the air entrainment rate by the amount equivalent to adding the strut.

*Examination of the Turbulence Model Effects*

It is apparent from the preceding case that the turbulence modeling approach impacts the cavity behavior and stability. At this point, it is useful to consider the physical mechanisms of air entrainment from the cavity as described in theory describing the importance of viscous interactions at the air-water interface. [89] Such viscous interactions pull on the cavity air forming air boundary layers that submit to the motion of the water. As the water progresses to the back end of the cavity, so does the air within these boundary layers. Eventually, this consists of the air that leaks from the cavity. A diagram displaying this behavior [89] is provided in Fig. 6-2. It is customary, as done in
turbulence models, to interpret turbulence as an effective viscosity; thus, the modeled turbulence is intimately connected to this air-boundary layer and ultimately the air entrainment rate.

Additional insight into the behavior of the turbulence model is gathered from the supercavitating body simulations. Reconsider the simulations from the supercavitating body (Case AC3), specifically at $C_Q$ of 0.45 where no hysteresis is present. Fig. 6-3 displays the predicted turbulence intensity field for the DES and URANS simulations at a similar cavitation number. Note the peak in the turbulence levels at the interface; however, the DES simulation has lower values than the URANS predictions. The increased turbulence levels imply an increased modeled eddy viscosity, thus, boundary-layer thickness. It appears that the general formulation of DES reduces this effect.
The effect of the turbulence model on these cavity boundary layers is further evaluated for the axisymmetric ventilated cavity (Case AC1). A high resolution near the interface is maintained in this study; the cell size in the cavity-normal direction is half of that of the fine grid used in the validation studies in Section 4.2.1. In this mesh, roughly sixty cells are used to resolve the cavity-interface boundary layers and the level-set method is used to remove as much of the interface diffusion as possible. The predicted boundary layers using RANS and laminar models are displayed in Fig. 6-4. The intention of the laminar case is to understand the limits of the possible solutions for the modeled cavity boundary layer behavior. The most obvious difference between the two cases occurs near the interface, which is best displayed at $x/R_N=20.8$. The RANS predictions show that the axial air motion is dominated near the cavity interface, or within the cavity boundary layers. In the laminar case, the air velocity quickly deviates from the higher
liquid velocity until reaching a roughly constant velocity region away from the interface. The integrated velocity in the cavity should be equal to the air-entrained and the ventilation rate. With the thinner laminar boundary layer having a consistent $C_Q$ with the turbulent case, the increased centerline velocity in the laminar case occurs to compensate for the decreased air entrainment through the boundary layers. In general, the amount of air entrained into the cavity-boundary layers is increased using a RANS model as compared to the laminar case.

![Graph](image)

**Figure 6-4**: Predicted cavity boundary layers for the axisymmetric disk cavitator (Case AC1) using laminar and RANS methods. This distance corresponds to the distance from the interface, into the air in the cavity. The velocity magnitude is used to account for the skewness of the cavity and the $\text{sign}(U)$ isolates entrained air from recirculating air in the cavity.

Based on the presented physical arguments, the expected sensitivities observed using various turbulence modeling approaches makes sense. In a DES modeling approach, the resolution and time-step sizes are too coarse for turbulent simulations of this cavity boundary layer. Whereas using a RANS model, the damping functions that
force the models to the appropriate behavior approaching walls are not used in the cavity-boundary layers. In the previously presented supercavitating body predictions, it is assumed that the DES simulations are fortuitously predicting the correct boundary layer behavior. These turbulence model deficiencies may be overcome via cavity-wall boundary-layer treatments within the turbulence model.

6.1.2 Natural Cavitation

The turbulence model approach is also investigated for natural cavitation problems. In this case, the twisted hydrofoil (Case VC1) is revisited and the solutions are examined with various turbulence modeling approaches. These studies are performed using the medium grid from the validation in Section 4.2.4 and all use a consistent time-step size. Recall from resolution studies, in Fig. 4-9, that this mesh and time-step size showed the ability to simulate the inertial range of the turbulence spectrum reasonably well, a requirement for such a comparison study. The examined turbulence modeling methods include URANS, DES, and MILES approaches. For these comparison studies, the time-averaged solutions are examined.

The time-averaged solution of the URANS simulation is displayed in several views in Fig. 6-5. An orthogonal view, in Fig. 6-5 (a), displays most the features of the predicted flow field. The red streamlines display streamlines passing over the wing, while the grey streamlines pass below. At a plane one chord length behind the wing trailing edge, vectors display the axially aligned vortical structures associated with the variable twist in the wing. The cavity shape is represented by the isosurfaces, which highlight the
channel structure predicted at the centerline. This channel region, or reentrant jet, creates a local pressure increase that reduces the amount of vapor. A view from the rear of the wing, with identical streamlines from the orthogonal view, is provided in Fig. 6-5 (b). The flow structure on the surface is displayed in Figs. 6-5 (c-d). These plots display the direction of the streamlines on the foil surface, which is the equivalent to oil-visualization techniques. The pressure-surface streamlines on the lower surface appear to be well aligned with the flow direction. The suction-surface streamlines behave as expected based on the previous views. An axially aligned flow is apparent outside of the cavity region. Within the cavity, near the leading edge, a recirculation region is apparent. The reentrant jet, occurring at the cavity centerline, has a length that is roughly equivalent to the chord length and displays an axially aligned behavior.
In Fig. 6-6, is the time-averaged solution of the unsteady DES prediction using similar plots as the URANS solution. The general flow field is similar to the URANS predictions as apparent in Fig. 6-6 (a); however, the predicted cavity shape differs significantly. The reentrant jet streamlines show a smaller portion of the jet aligning with the axial flow direction than was observed in the URANS prediction. There are also smaller eddies forming at the edges of the reentrant jet that are not predicted when using the URANS approach. The reentrant jet from the DES simulation also appears to elevate
higher from the wing surface and extends further along the span than the URANS predictions. In Fig. 6-6 (c), is the top view of the foil; in this case, the mean length of the predicted jet is roughly 0.5c as compared to 1.0c for the URANS case. The flow structure on the foil surface also differs from the URANS prediction; the recirculation zones tend to be smaller and spread more along the span. Furthermore, in the wake of the reentrant jet a span-wise flow is predicted that is not apparent in the URANS prediction.

Figure 6-6: Time-averaged solution views of the twisted foil (Case VC2) using an unsteady DES approach. The wing is colored by $c_{pb}$, the red streamlines pass over the suction surface, the grey streamlines pass over the pressure surface, the blue translucent isosurface is at $\alpha=0.5$, and the grey isosurface is at $\alpha=0.1$, and the black lines are surface streamlines.
It is important to understand the regions where the DES approach modifies the turbulence model behavior. In Fig. 6-7, some insight of the locations of the LES regions are observable in the contours of the time-averaged values of $F_{DES}$ at various spanwise locations along the span. The LES-like simulation appears to affect the vaporous regions near the wing centerline, where the contours slices are redder. At span-wise locations that approach the walls, where no vapor is present, the disappearance of the contour plots indicates that $F_{DES} \rightarrow 1.0$ and the baseline turbulence model is recovered.

![Figure 6-7](image.png)

Figure 6-7: Time-averaged values of the DES switch, $F_{DES}$, for the twisted, cavitating wing (Case VC1) at various span-wise locations. The blue translucent isosurface is at $\alpha=0.5$, and grey at $\alpha=0.1$.

A time-averaged solution of the unsteady MILES prediction, using the now familiar plots from the DES and RANS solutions, is presented in Fig. 6-8. Note that a consistent mesh and time-step size are used across all cases. In comparing these predictions to those from the URANS, the differences parallel those between the DES and URANS solutions. Small differences between the DES and MILES solutions exist, primarily in the predicted reentrant jet. The recirculation regions of the MILES prediction are significantly larger than predicted in the DES case. This larger jet appears to shorten
the high-vapor-content region of the cavity, indicated in blue, to shorter lengths than observed in the RANS- and DES-simulation solutions. This increased span-wise reentrant-jet behavior should be expected as, in general, turbulence models tend to dampen the simulated turbulent structures. With the decreased modeling of turbulent flow, using the MILES model, as expected, the structure-size increases. Lastly, the boundary layers are estimated to be transitional for this hydrofoil, which is based on the analysis performed in Appendix B.6. A MILES simulation provides an improved approach for laminar boundary layers, whereas DES and URANS better model turbulent ones. The impact of this is not yet fully understood.
A comparison of the predictions using various turbulence models and the experiments is given in Fig. 6-9 from a view looking on suction surface of the wing. The experimental photograph is generated from a combined, or averaging, of the images of a video sequence [74] using ImageMagick®[90]. Such a representation provides a better measure for these comparisons. The reader is referred to Fig. 4-12 for visualization of the still frames that are used to generate this averaged representation. The red-dashed lines outline where the cavity shape appears in the experimental photograph, and the yellow distinguishes the phase-separated and foamy parts of the cavity. The lines are also placed

Figure 6-8: Time-averaged solution views of the twisted foil (Case VC2) using a MILES approach. The wing is colored by $c_{p,b}$, the red streamlines pass over the suction surface, the grey streamlines pass over the pressure surface, the blue translucent isosurface is at $\alpha=0.5$, and the grey isosurface is at $\alpha=0.1$, and the black lines are surface streamlines.
on the CFD predictions for comparison purposes. The CFD simulations are all time-averaged representations. Two isosurfaces at vapor-volume fractions of 0.7 (dark blue) and 0.3 (light blue) are used to represent the phase-separated and foamy regions, respectively.

![Figure 6-9: Comparison of the simulations using various turbulence model approaches to an experimental video [74] compressed into a single image. The cavity from the experiment is outlined by the red-dashed line and the mixture region is indicated using the yellow-dashed line. The dark-blue and light-blue isosurfaces are at $\alpha=0.7$ and $\alpha=0.3$, respectively.](image)

In the vaporous regions pictured in Fig. 6-9, the methods that attempt to simulate the inertial-range turbulence (i.e. DES/MILES) seem to weaken the coattail-like cavity structures and increase the cavity size along the centerline. Note that the outline from the combined-experimental video is overlaid on the predictions. Comparing the general cavity shapes, the cavity size varies significantly using all approaches of handling the turbulent flow. All of the predictions show cavity features that tend to align well with the outline. The major deviation for the URANS and DES simulations is in the predicted coattail structures. The MILES simulations displays damped coattail structures, but the core of the cavity appears quite large. The coattail-like structures that form in the DES
and URANS predictions present a concern. Small amounts of vapor are observed in the videos in this region, but the amount predicted is clearly too high. Finally, all of the simulations fail to predict the portion of the cavity that extends in the spanwise direction at the leading edge.

Similar studies have been performed for the same experiment and conditions comparing LES and URANS predictions [91]. Relative to the present study, those of Huuva [91] were performed on slightly higher-resolution meshes. A notable difference is the rather dynamic URANS prediction compared to a nearly stationary URANS prediction on a coarser mesh used in these results. Note that a different cavitation model was also used in the URANS simulations of Huuva [91], which may also indicate a cause of the more dynamic URANS predictions. In general, similar conclusions are observed, where the simulation of high-frequency cavity dynamics is revealed using a LES approach as compared to URANS. One major advantage of the DES approach, over the LES approach, is for applications to high-Reynolds number flows. For example in this twisted wing case, having a Reynolds number of 1.146×10⁶, the predictions from Huuva [91] clearly display an unlikely separation region that none of a DES, RANS, or XFOIL predictions (see Appendix B.6) display over the angle of attack range bounded by the twist in the wing. This premature separation is indicative of the inability to capture turbulent separation, probably due to lack of temporal and spatial resolution of the unmodeled boundary layer turbulence in the LES formulation. These issues are alleviated using a DES approach by modeling these high-frequency scales.

The time-averaged force predictions using the various turbulence modeling approaches are provided in Table 6-1. Note that the predictions improve with the added
simulation of the turbulent behavior. However, these differences from the experimental measurements are still significant. It is expected that improved meshes and fewer assumptions will improve the predicted lift. Some likely problematic assumptions are the inflow, where a specified velocity rather than total pressure is used. This type of boundary condition is important for internal flows. Compressible effects also need to be ruled out and are performed later in the chapter. Furthermore, tunnel wall viscous effects may also be affecting the study. Such studies still need to be investigated for a proper validation.

### Table 6-1: Lift predictions compared to measurements with varied turbulence models

<table>
<thead>
<tr>
<th>Model</th>
<th>$C_L$</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>URANS</td>
<td>0.397</td>
<td>-23.9</td>
</tr>
<tr>
<td>DES</td>
<td>0.427</td>
<td>-18.2</td>
</tr>
<tr>
<td>MILES</td>
<td>0.446</td>
<td>-14.4</td>
</tr>
<tr>
<td>Experiment</td>
<td>0.522</td>
<td>-</td>
</tr>
</tbody>
</table>

### 6.1.3 Oscillating, Cavitating Wing

Similar studies examining the turbulence modeling approach for the oscillating, cavitating wing geometry (Case VC3) are performed. The experimental cavity is compared to that predicted using the URANS, DES, and MILES turbulence models, which is displayed in Figs. 6-10 and 6-11. The displayed instances can be referenced in the load prediction plots in Fig. 6-12 using the vertical-dashed lines. Note that there is no way to correspond the solutions with the experimental photographs in time. Thus, the photographs are selected that best represent the predicted URANS cavities.
Instances from the first cavity shedding cycle are displayed in Fig. 6-10. As the cavity develops, in Fig. 6-10(a), a similar stable cavity is predicted using all of the methods. The predicted cavities all agree with those observed in the experiments; this agreement includes the general shape, the cavity curvature approaching the tip, as well as the cavity interacting with the water-tunnel wall at the fin-wall junction. The initial destabilization of the cavity occurs as it grows past the hydrofoil trailing edge and is apparent in Fig. 6-10(b). With all of the turbulence models this destabilization behavior is well predicted; however, using the MILES model this feature is predicted to slightly lag that of the others. Note that at the point where the fin stalls, or lift breakdown, is at a time just after this destabilization. The initial unstable cavity in Fig. 6-10(c) is similar between all methods.
Figure 6-10: Comparisons of predictions (with turbulence model variations) and experiments during the initial cavity formation and destabilization in a hydrofoil oscillation. The columns, from left to right, respectively display the cavities observed in experiments [77], URANS, DES, and MILES turbulence model predictions. The pink and white-translucent isosurfaces represent the $\alpha = 0.5$ and 0.1 values, respectively.
The initial cavity-shedding event is displayed in Fig. 6-10 (d-e). In this stage, only minor differences occur between the solutions, and all agree well with the cavities observed in the experiments. Perhaps the only distinguished feature is the predicted hairpin vortex in the RANS solution, which is significantly smaller than that predicted in the DES and MILES solutions. The vortical feature in question appears in Fig. 6-10(e). In general, the stable cavities display little to no influence from the turbulence model. Only when the cavity destabilizes and cloud cavities are shed do the cavity predictions vary.

The secondary shedding cycles are displayed in Fig. 6-11. In a general comparison, the RANS solutions display less cavity structure than both the DES and MILES solutions. However, comparing the DES and MILES predictions, the apparent cavity structure is indistinguishable. This behavior suggests that the modeled eddy viscosity in the DES simulations, for this particular mesh, damps the cavity dynamics to an amount similar to that of the MILES formulation. Such an observation implies that, provided that the mesh resolution is sufficient, the modeled dissipation in the LES and DES formulations impacts the cavity in a similar manner. Again, the general characteristics of the predictions are in agreement with the experiments for all simulations.
Figure 6-11: Comparisons of predictions (with turbulence model variations) and experiments during the secondary cavity cycle and the recovery from the cavitation event in a hydrofoil oscillation. The columns, from left to right, respectively display the cavities observed in experiments [77], URANS, DES, and MILES turbulence model predictions. The pink and white-translucent isosurfaces represent the $\alpha = 0.5$ and $0.1$ values, respectively.
The predicted loads are compared in Fig. 6-12. In this study, the hydrodynamic predictions are not in good agreement with the experiments. This is expected, as the modeled oscillation frequency is higher than the experiments, which is examined in detail later in Chapter 6. In this section, only the effects of the turbulence modeling are studied with respect to the predicted hydrofoil loads. The lift, drag, pitching moment about the quarter chord, and the root-based flap-wise bending moment are all plotted against the nondimensional time in Figs. 6-12 (a), (b), (c), and (d), respectively. Note that the modeled pitch cycle is displayed with a constant multiplier. For the lift, this constant is based on the three-dimensional lift-curve slope based on lifting-line theory.

The predicted load responses between for the various turbulence models display only slight mean-load variations. The high-frequency cavity dominated load response predicted with the DES and MILES methods has an increased amplitude and a decreased frequency compared to the URANS prediction. These features are an effect of the varied prediction of the large-scale vortical-cavity structures observed in the solutions. For example, refer to Fig. 6-10 (e), where the hairpin vortex of the URANS solution has dissipated compared to the DES and MILES predictions. The effect of this dissipation lowers the intensity of the vortex core, decreases the effect of the pressure reduction in the core, and decreases the time related to this large-scale cavitating vortex. This time reduction of the cavitating scales corresponds to the increased frequency in the cavity-induced loads predicted with a URANS approach.
The sensitivity of the turbulence models on the unsteady, cavitating loads is displayed in Fig. 6-13 (a) using the lift-power-spectral densities (PSD) in the frequency domain. Note that a representation of the pitch cycle (CFD: Pitch) is also provided. This equivalent lift-power spectrum is based on the quasi-steady, single-phase lift plotted in Fig. 6-12 (a) (listed as \( \text{CONST}\bullet \alpha \)). Here, a combination of the simulated angle of attack and three-dimensional lifting-line theory enable such a comparison, and is useful to
understand the deviation from the single-phase conditions and how the pitch cycle influences the predictions. Using these comparisons, an evaluation of the effects of the models on the unsteady loads can be made. In Fig. 6-13 (b) are comparisons of simulations with the experimental data, again using the lift PSD. For a better comparison, the CFD lift is sampled at the same rate that the lift measurements of the experiments were. Recall that the simulated pitch frequency is different that was performed in the experiments. This makes it hard to compare with $c_L$-$\alpha$ or $c_L$-time curves, but the lift-PSD plots should remain relevant at the frequencies where cavitation dominates the loading power. These regions are apparent where there is deviation in the pitch (CFD: Pitch & Exp: Pitch) and computed lift curves. The predictions, using any of the turbulence models, display agreeable predictions with the experiments.

Figure 6-13: Power spectra of the lift loads on the fin from predictions using various turbulence models and compared to experiments. The pitch cycle from the CFD simulation (CFD: pitch) displays the power spectra based on three-dimensional lifting-line theory and the simulated pitch cycle. Similarly, the experimental pitch cycle (Exp.: pitch) extends the pitch cycle from the experiments to a lift-power spectra curve.
The effects of the turbulence models on the predictions can be further understood in the predicted power spectra, in Fig. 6-13 (a), which are compared to equivalent lift power-spectra of the pitch cycle. The predicted responses using all of the turbulence models display increased loading power at the moderately high frequencies, $0.2 < f_c/U < 2$, when the lifting surface is cavitating. It appears that the loads in the RANS solution lock into the pitch cycle at roughly $f_c/U > 1$. Whereas the DES and MILES cases sustain a higher lift PSD, until about $f_c/U = 5$, and then drop to levels below the pitch cycle. Furthermore, both the DES and MILES cases display independence from the high frequency pitch cycles. Such behavior indicates that the high frequency, turbulent loads are being simulated.

6.1.4 Summary of Improved Turbulence Models

Improved turbulence models that include the addition of turbulence simulations show some promise in improving the prediction of cavitating-fluid flows. In general, no effect was observed for steady-cavitating flows. For artificial supercavitation, the approach is crucial, as baseline RANS and URANS approaches amplify the amount of air entrained from the cavities. In these cavities no benefit was displayed until hysteresis was observed, where DES improved the prediction capability. For unsteady and partially cavitating flows these turbulence modeling improvements displayed a better overall prediction of cavity dynamics. However, the benefits are not significant. In general, the approach displays moderate to significant improvement in the predictions, but proves to be case dependent.
As previously discussed in Chapter 1, the bulk speed of sound in high-density ratio multiphase mixture introduces a need to consider highly compressible flows. This compressible feature is a result of an altered density-pressure response. A pressure perturbation in a multiphase flow creates a rather small density changes to all of the fluids; a linearized analysis implies that the density changes of each species correspond to their sound speed. The significant sound-speed reduction is a result of the subsequent volume-fraction change in a mixture with a varied density-pressure response amongst the species. Thus, in a high-density ratio mixture with varied sounds speed a small pressure perturbation can significantly alter the mixture-density. Such density-pressure relations physically act to reduce the sound speed. As in gas-dynamics problems, the behavior is also hyperbolic, but the fluid properties and temperature jumps across shocks are contrarily quite small in these gas-water mixtures. For this reason, the isothermal compressible assumption is often invoked implying that a linearized density-pressure response can still represent the delayed pressure response, and is given as

\[ \rho \approx \rho_0 + (p - p_0) \frac{\partial \rho}{\partial p} + (T - T_0) \frac{\partial \rho}{\partial T} \approx \rho_0 + \left( \frac{p - p_0}{a} \right). \]  \hfill 6.1

Using this equation, the need to solve the energy equation is eliminated. However, gas-dynamics problems remain of interest in the application of rocket propulsion plumes and ventilation systems. For this reason, the ideal-gas law or a stiffened-ideal-gas law are sometimes used. The stiffened-ideal-gas law is written as

\[ \rho = \frac{P + P_{\text{Stiffen}}}{RT}. \]  \hfill 6.2
where the $p_{\text{stiffen}}$ term is used to adapt the law to approximately model the water. When $p_{\text{stiffen}}$ is zero, the ideal gas law is recovered. Other state equations can be developed based on experimental data fits. For example, consider

$$\rho \approx \rho_0 + (p - p_0) \frac{\partial \rho}{\partial p} + (T - T_0) \frac{\partial \rho}{\partial T} \approx \rho_0(p_0, T) + \frac{p - p_0}{a(p_0, T)^2}, \quad 6.3$$

where both $\rho_0$ and $a$ are functions of the temperature that are interpreted at a constant pressure, $p_0$. Such an equation is formulated like the isothermal state equation, buts adds a degree of freedom that incorporates the density-temperature relationship. The form to the right is preferred as it is a simple extension of Eq. 6.1. As the pressure deviates from $p_0$, small errors tend to occur in the temperature behavior. However, the density is an explicit function of pressure and temperature, which provides a more efficient state relation that can still accurately represent complicated regimes. The typical assumption is to base these relations at the saturation pressure as a function of temperature. With such a model, thermally driven compressible effects associated with phase change can be captured accurately and efficiently along the vapor dome.

There are several algorithm capabilities required for performing these compressible, multiphase computations. One of the more important capabilities is the all-Mach number formulation, which enables the simulation over the wide range of Mach numbers encountered in such flows. As previously mentioned, the free stream Mach number is typically low and yields a poorly conditioned linear system for a standard compressible-flow solver. In cavity mixtures, however, the Mach number is generally high and presents a case that is poorly suited for incompressible solvers. Thus, an all-Mach number scheme is really needed to efficiently simulate these cavitating flows.
Next, having the flexibility in the state equation is required to model the fluid properties of all species. These features of UNCLE-M, in particular, enable such evaluations.

### 6.2.1 Predicted Effects of Compressible Fluids

The effects of modeling compressible mixtures are evaluated in several three-dimensional partially cavitating flow calculations. The mixture-fluid regions are where the compressible effects are expected. In the phase-separated regions, i.e. regions separated into all water or all gas, the flow is expected to behave like an incompressible fluid.

#### 6.2.1.1 Partial Cavitation: Twisted Hydrofoil

The first case examined is the cavitating twisted wing geometry (Case VC1). These cases all use the DES turbulence modeling approach. In Fig. 6-14, are the time-averaged solutions using an isothermal-compressible prediction. The time-accurate solutions for this case can be referenced in Figs. 3-12 and 3-13. Recall that in the isothermal-compressible prediction the density is modeled as a function of pressure and not temperature. For comparison purposes, identical solution views, mesh, and time-step sizes as in the turbulence modeling studies are used. An understanding of the impact of compressible model formulations can be extracted by comparison to the incompressible DES case. Between these cases, significant changes in the cavity shape compared to the incompressible cases in Fig. 6-6 are present. The top view is the better view for the
comparisons. In this compressible case, the coattail structures are greatly reduced. Furthermore, the recirculation region within the cavity shows a similar pattern, with a much larger size than the incompressible prediction.

In Fig. 6-15 is a similar plot with a fully compressible solution method. Within this model the full-energy conserving equations are solved, the vapor-pressure-temperature relations are included, or \( p_v(T) \), and the water-based temperature effects are considered. A significantly different cavity is predicted than even the isothermal-compressible predictions. Here, the predicted cavity is shorter and extends further in the
spanwise direction, the coattail structures are gone and the predicted separation pattern has again changed. This is the most inclusive model, thus, is expected to be the most physically accurate. Note that the thermal-water-properties effects tends to limit the stable time-step size and Courant number values of less than unity are required for stable and/or accurate integration. This implies an increased computational effort for the same physical-time period modeled when compared to the simulations using the other state relations.

Figure 6-15: Time-averaged solution views of the twisted foil (Case VC2) using a DES, fully compressible approach. The wing is colored by $c_{p,b}$, the red streamlines pass over the suction surface, the grey streamlines pass over the pressure surface, the blue translucent isosurface is at $\alpha=0.5$, and the grey isosurface is at $\alpha=0.1$, and the black lines are surface streamlines.
In Fig. 6-16, is a comparison of time-averaged solutions with variations in the state-equation assumptions. As before, this is view looking on suction surface of the wing that compares the predictions to an experimental photograph generated from a combined, or averaging, of the images of a video sequence [74] using ImageMagick® [90]. Such a representation provides a better measure for these comparisons. The red-dashed lines outline where the cavity shape appears in the experimental photograph, and the yellow distinguishes the phase-separated and foamy parts of the cavity. The lines are also placed on the CFD predictions for comparison purposes. The CFD simulations are all time-averaged representations. Two isosurfaces at vapor-volume fractions of 0.7 (dark blue) and 0.3 (light blue) are used to represent the phase-separated and foamy regions, respectively. The main cavity shape is predicted well using any of the assumptions in the state relations. However, the main cavity structure is best predicted using the fully compressible assumption, where the predicted cavity finally displays the spanwise extensions. This spanwise lengthening of the cavity is only apparent when the water-property temperature dependencies are included in the model. Although the fully compressible cavity may appear shorter than the experiment, it is expected that the \( \alpha=0.1 \) isosurface may not align with the visually observed cavity. Perhaps a lower volume fraction would align well with the experimental profile. The incompressible-fluid cavity prediction displays coattail-like structures that are only weakly apparent in the experiment. It appears that the thermodynamic models influence the prediction of these coattail structures, where the fully compressible model completely eliminates them.
In Table 6-2 are the predicted lift values compared to the experiments with the varied thermodynamic models. It is apparent that the predicted lift is still significantly less than that measured in the experiments. At this point, only mesh and more inclusive models of the walls and boundary conditions are expected to improve the predictions.

<table>
<thead>
<tr>
<th>Model</th>
<th>( C_L )</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Incompressible</td>
<td>0.397</td>
<td>-23.9</td>
</tr>
<tr>
<td>Isothermal Comp.</td>
<td>0.426</td>
<td>-18.4</td>
</tr>
<tr>
<td>Full Comp.</td>
<td>0.400</td>
<td>-23.4</td>
</tr>
<tr>
<td>Experiment</td>
<td>0.522</td>
<td>-</td>
</tr>
</tbody>
</table>

6.2.1.2 Partial Cavitation: Oscillating Hydrofoil

The effects of the state equation model and assumptions are examined for the oscillating, cavitating wing geometry (Case VC3). Comparisons to the experimentally
observed and predicted cavities using incompressible, isothermal-compressible, and full energy-conserving-compressible models are displayed in Figs. 6-17 and 6-18. As was done in the turbulence model comparisons, each displayed instance is sequentially indicated in the load prediction plots in Fig. 6-19 with the vertical-dashed lines.

The first cavity cycle is displayed in Fig. 6-17. The cavity develops with a stable pattern and the predictions are compared in Fig. 6-17 (a); as before, all methods predict similar cavities. Furthermore, all of the predicted cavities agree with the cavity observed in the experiments. This agreement includes the general shape, the cavity curvature approaching the tip, as well as the cavity interacting with the water-tunnel wall at the fin-wall junction. The destabilization of the cavity initiates at the trailing edge, as displayed in Fig. 6-17 (b), where all cases are agreeable with each other and the experiment. The early unstable cavity, shown in Fig. 6-17 (c), starts to show deviation between the cases with different approximations. The isothermal compressible prediction displays a larger vortical structure than both the incompressible and fully compressible cases, however, it is likely that the cavity-shedding cycle in the isothermal compressible case is leading that of the cycle in the other cases. This feature is observed in Fig. 6-17 (c-e). The formation and shedding of the vortical structures in the fully compressible prediction appears to lag both of the other solutions. In this case, the stable cavities display little to no dependence on the modeled state equation behavior, however, differences are still observed as the cavity becomes unstable.

The secondary shedding events and the recovery from the cavitation event is displayed in Fig. 6-18. All cases display many cavity scales, and in general, agree well with the experiments. There are obvious differences in the displayed solution that is
likely a combined effect of the sampling instance and the varied time lags of the predictions. The shedding behavior is perhaps better analyzed by comparing integral measures.
Figure 6-17: Comparisons of state equation assumptions and experiments during the initial cavity cycle. The columns, from left to right, respectively display the cavities observed in experiments [77], incompressible, isothermal compressible, and full energy conserving compressible with $p_v(T)$. All cases use the DES turbulence model. The pink and translucent white isosurfaces represent the $\alpha = 0.5$ and 0.1 values, respectively.
Figure 6-18: Comparisons of state equation assumptions and experiments during the secondary cavity cycles and recovery. The columns, from left to right, respectively display the cavities observed in experiments [77], incompressible, isothermal compressible, and full energy conserving compressible with $p_s(T)$. All cases use the DES turbulence model. The pink and translucent white isosurfaces represent the $\alpha = 0.5$ and 0.1 values, respectively.
Comparisons of the predicted loads are provided in Fig. 6-19. These compare the loads from the previously displayed simulations. The lift, drag, pitching moment, and flap-wise bending moment at the root are all plotted against a nondimensional time in Fig. 6-19 (a), (b), (c) and (d), respectively. As previously mentioned, the modeled pitch-angle cycle is displayed with a constant multiplier to indicate the pitch cycle, where for lift this is based on lifting-line theory. The mean response of the solutions agree well with each other. Differences are observed in the unsteady force response that shows slightly altered high-frequency loads associated with the cavity dynamics. The isothermal-compressible case displays a slightly higher frequency than the others. Whereas the loads on the fully compressible case appear to lag those of the other solutions, which is consistent with observations made in Fig. 6-17.
The PSD of the lift is plotted against frequency in Fig. 6-20 (a). In general, the trends of all predictions agree well with each other up to about \( fc/U = 2 \). Following that point, the lift from the fully compressible simulation locks into the pitch cycle. The other cases, however, deviate significantly. This indicates that some of the cavity dynamics do in fact correlate with the high-frequency pitch oscillations. As before, the experiments and CFD lift predictions are sampling at similar frequencies compared in Fig. 6-19 (b).
All methods predict roughly the same PSD as the experiments in the range where the lift is dominated by cavitation.

6.2.2 Summary of the Compressible-Flow Modeling Effects

Modeling cavitating-fluid flows as compressible fluids can affect the predicted flow field. Similar to the turbulence models, the was little impact when the cavities were stable. For unsteady and highly dynamic cavities, the fluids behave more compressible and modeling those effects becomes important. In particular, predicting the correct cavity shape correlates with modeling temperature-varying properties of each species. The approach can lead to significantly longer simulation efforts, thus, may not be desirable to capture in each case. Regardless, modeling the correct thermodynamics remains an
important physical behavior that needs to be considered when modeling cavitating-fluid flows.

6.3 Summary

For the cases examined, the simulation of turbulence and modeling of compressible features appear to improve the simulation fidelity. The examined cases were partially cavitating, unsteady flows that are expected to be more troublesome to predict than largely steady cavities. With the simulation of turbulent features and full energy conservation, features in the predictions converge towards the experiments. Although cavity shape and dynamics are predicted well, issues remain in predicting unsteady lift of the partially cavitating wing. These studies only briefly examine the more stable cavity type for an oscillating wing, where both of these physical modeling improvements were insignificant. In general, these modeling capabilities may not always be required for accurate simulations, but the effects suggest an initial strategies for improving predictions.
Chapter 7
Computational-Based Investigations of Physical Behavior

After validating the overall approach, it is reasonable to evaluate the physical behavior of the predictions with confidence. Extracting such information is most useful to understand behavior that is difficult for experimental methods to extract. Although extracting physical behavior from CFD simulation is not necessarily verification of physical behavior, in some cases it is more efficient and effective than examining the complex physical phenomena experimentally.

7.1 Numerical Examination of Cavitating Flows

7.1.1 Hysteretic Cavity Prediction

Hysteresis is a common feature in cavitating flows that is both advantageous and troublesome for cavitating vehicle design. The predictive ability of such behavior is important to understand and analyze the operation envelope of a stable cavity. In the hysteretic range, the cavities have a tendency to remain at their original state; thus, for a given $C_Q$, a shrinking cavity can be larger than a growing cavity. In essence, situations arise causing the cavity size to be path dependent.

The prediction of such hysteretic behavior is illustrated in the $C_Q$-$\sigma$ curve for the supercavitating body, at various ventilation rates, (Case AC3) in Fig. 7-1(a). Note that
these solutions use a DES approach and assume a symmetry plane; it was shown in Section 6.1.1 that the support strut aligns the prediction to the experiment data. In such a hysteretic flow, the effect of the initial condition on the predicted solution is also displayed and is a dominant factor in the predicted solution. For this specific case, the impact of the initialization approach affects the cavity size as well as the corresponding vehicle loads.

In further inspection of Fig. 7-1, several regimes based on the behavior of the numerical predictions are observed. In general, for high gas flow rates ($C_Q>1.7$), this specific configuration displays a large high-pressure twin-vortex-closing cavity. Note that the predictions in this regime are omitted to highlight the hysteretic region. At lower gas ventilation rates ($C_Q<=1.0$), a small low-pressure reentrant cavity is predicted. When the $C_Q$ values are in the hysteresis regime ($0.1< C_Q <0.3$), only small changes in the cavity

![Figure 7-1](image)

(a) $C_Q-\sigma_c$ Comparison  
(b) Corresponding air recovery

Figure 7-1: Predicted hysteretic cavity behavior compared to experiments [12] of a ventilated supercavitating body (Case AC3). (a) Comparisons of the experimentally obtained and computed $C_Q-\sigma_c$ curves revealing hysteresis and the (b) corresponding boundary layer air recovered, $k_{REC}$.

In further inspection of Fig. 7-1(a), several regimes based on the behavior of the numerical predictions are observed. In general, for high gas flow rates ($C_Q>1.7$), this specific configuration displays a large high-pressure twin-vortex-closing cavity. Note that the predictions in this regime are omitted to highlight the hysteretic region. At lower gas ventilation rates ($C_Q<=1.0$), a small low-pressure reentrant cavity is predicted. When the $C_Q$ values are in the hysteresis regime ($0.1< C_Q <0.3$), only small changes in the cavity
pressure, thus size, occur until the hysteretic region is breached. At that point, a jump in the cavity size and pressure occurs. The ability to capture such phenomena goes a long way towards establishing CFD as a credible tool for modeling the physics of supercavitation. Interestingly, the curves remerge at the point where the RANS-based solution tended to lose the large cavity solution. Thus, as mentioned earlier, it is concluded that the URANS methodology does not appear suitable where these multivalued regimes exist.

The physical cause of the hysteresis is best described as a geometrically driven one. As discussed by Semenenko [4], such behavior is expected to occur over bumps or appendages on a supercavitating body. In this case, the hysteretic behavior occurs over the shoulder of the body, or the corner forming at the initial point of the parallel midbody. Plots of the predicted cavity shape at various ventilation rates with different initial conditions are displayed in Fig. 7-2. As implied by the $C_Q-\sigma_c$ curve, in Fig. 7-1 (a) and (b) the cavity size appears to be inversely proportional to $\sigma_c$. Cavities from the decreasing $C_Q$ predictions are presented in Fig. 7-2 (a), showing a strong tendency to stay over the shoulder. For the increasing $C_Q$ case, in Fig. 7-2 (b), the predicted cavities at low ventilation rates remain ahead of the shoulder. Eventually, the ventilation rate is increased to the point where the shoulder is breached and the cavity length quickly increases. The cavity length response with respect to the ventilation rate has a relationship that is quite similar to the example presented in Semenenko [4], and is displayed in Fig. 7-2 (d). This gives further confidence that the hysteresis is physically accurate and that a CFD approach can predict such behavior.
Figure 7-2: Predicted cavity shape, length, and corresponding ventilation rate through the hysteresis loop of the supercavitating body case (Case AC3). Note that the predicted cavities are displayed with the $\alpha=0.8$ isosurface, and the black, purple, blue, red and pink cavity shapes are for $C_Q=0.08, 0.2, 0.32, 0.45,$ and $1.0$, respectively.
The predicted cavity shape is overlaid on the experimental photographs \([12]\), at various rates of ventilation, in Fig. 7-3. The cavity shapes for both the low- and high-\(\sigma_c\) CFD predictions are displayed. From the available data, only qualitative comparisons are possible. This is sufficient to examine the behavior of the predicted cavity in comparison to those produced in the experiments. In general, the smooth front part of the cavity is well predicted for all of the low-\(\sigma_c\) solutions. The predicted wake and mixture regions of the experiments slightly vary from those predicted. This is partly due to the ambiguity of defining an equivalent isosurface, possible breakdown of the homogenous velocity assumption in this region, and the neglected support strut in this particular simulation. Regardless, the prediction of this reentrant-type cavity shows promise in the validity of integrating CFD into the analysis process.
Figure 7-3: Comparisons of predicted cavity shapes to experimental photographs of the supercavitating body (Case AC3). The photographs in the upper part of each subfigure are borrowed from the experiments of Schauer [72], and the center part of each subfigure is the high-σ solution, and the lower part of each subfigure is the low-σ solution.
7.1.2 Mechanisms of Air Entrainment from a Ventilated Cavity

Theory of gas entrainment from cavities with toroidal-vortex closure has been based on the concept of gas exiting from these toroidal-shaped, shedding cavities. Recent theory suggests that mechanisms occurring upstream of these toroidal vortices are important, and that the gas transported along interfacial shear layers to the end of the cavity consists of the air that eventually fills the toroidal cavities [89]. Recall the diagram in Fig. 6-2, where the concept of the air-cavity shear layers entraining air from the cavity is described. This concept of cavity-shear layers dragging air from the cavity was previously hypothesized, as mentioned in May [13], however, was never really developed into a complete air entrainment model. The theory presented by Spurk [89] shows an excellent prediction capability when compared to experimental measurements for high-Froude number, reentrant cavities with little buoyant effects.

It is suggested that this theory, thus far applied to slender, reentrant, axisymmetric supercavities, is applicable to buoyant ones as well. Based on inspections of the previously validated simulations, the turbulence modeling sensitivities, and the success of the finite volume method itself, the concepts of Spurk [89] are further supported. The theoretical model proposed by Spurk [89] for the air entrainment rate, is given as

\[ C_0 = k_Q \frac{(1+\sigma_c)}{\sigma_c} \sqrt{\frac{1}{\sigma_c} \ln \frac{1}{\sigma_c}}, \]  \hspace{1cm} (7.1)

where \( k_Q \) is an empirically determined constant that is based on a single reentrant cavity data point. Note that other forms of this entrainment model based on laminar or turbulent boundary layers are presented, however, this empirical form is used as it calibrates well...
to any cavity. When Eq. 7.1 is applied to the reentrant-cavity curve of the supercavitating body [72], in Fig. 7-4, the predicted $\sigma_c$ values using the RANS CFD approach agree better with the theory, rather than the experiments. It is expected that without the presence of the body, this particular form of hysteresis would not exist and the cavity behavior would correspond to the theoretical behavior.

The streamlines displaying the airflow of the supercavitating body are shown in Fig. 7-5 (a). The red streamlines, seeded aft of the vehicle from the entrained air, originate directly from the ventilation ports or air entrained from the cavity, but still along the cavity shear layers. The purple streamlines are seeded within the cavity and show that the cavity air remains as an isolated region. These streamlines clearly display the cavity boundary layers entraining air from the cavity, which is the exact mechanism described in the work of Spurk [89]. In Fig. 7-5 (b), contours at various axial locations and along the symmetry plane are colored by the local air-entrainment rate. The local air

Figure 7-4: Examination of the experimental results of a supercavitating body (Case AC3) [72], and the RANS and URANS CFD predictions compared to the Spurk’s theory.
entrainment rate, $Q_{LOC}$, is defined as $Q_{LOC} = \alpha_{\text{air}} \frac{u}{V_{\infty}}$. Note that only positive values, or the air that is being entrained, is colored. With the exception of the ventilation ports, the positive values correspond well with the cavity and further verify the theory of Spurk.

Figure 7-5: The $C_D=0.45$ RANS prediction of the ventilated, supercavitating body (Case AC3). The contours display the local air entrainment, $Q_{LOC}$, and the streamlines indicate the path of the entrained versus cavity air. Note that little air escapes the cavity region and the predictions display air entrainment along cavity shear layers.
The origin of the recirculating cavity air at various locations near the cavity terminus is displayed in Fig. 7-6 for the twin-vortex-closing supercavitating cone cavitator (Case AC2). The contours at various axial locations are again colored by the local air-entrainment rate. The streamlines are colored by a dimensionless pressure gradient and the symmetry plane is colored by the volume fraction where the cavity is outlined in black. Figure 7-6 (a) displays the origination of streamlines seeded at the minimum $Q_{LOC}$ value in the next to most-aft plane. Such visualization is useful as it indicates the origin of the recovered air. The streamline history of this recovered air shows entrainment into the cavity boundary layer; then, near the cavity terminus, the outer portions of the boundary layer separates at the corresponding locally positive pressure gradient. In Fig. 7-6 (b), the streamlines are seeded close to the regions where $Q_{LOC}$ is only slightly negative, or a small amount of air is recovered. These streamlines display a similar behavior to the previous streamlines, that is, they contain boundary layer entrained air that later is separated from the cavity-wall shear layer. The major difference is the history of the streamlines, where these originate from both the cavity and the ventilation port. Examining the history of the streamlines seeded at the maximum $Q_{LOC}$ values, in Fig. 7-6 (c), all originate directly from the ventilation ports. The author is unaware of air entrainment theory that accounts for these viscous air-entrainment mechanisms in a twin-vortex cavity.
Considering the observed mechanisms of air entrainment, it is not straightforward to model all the physical processes occurring. For example, Spurk [89] focuses on air in the cavity-boundary layers that is pulled to the aft end of the cavity and is all eventually entrained. Other theories, namely for twin-vortex cavity closure, suggest that the core of the trailing vortices sustains a similar pressure as in the cavity. Note that deviations from

Figure 7-6: Air entrainment and recovery for twin vortex cavity on a cone cavitator (Case AC2). The streamlines (colored by pressure gradient) are traced backwards to indicate the origin. The contour lines display the local air entrainment rate, $Q_{LOC}$. The background is a contour plot of the air volume fraction with black indicating air.
this model are required for low Froude number conditions. In the work of Campbell and Hilborne [92], a theoretical formulation of the air entrainment is based on the computed circulation about the cavity centerline that corresponds to the buoyant loads on the cavity and correlates to the radius of the vortex core. A resulting air entrainment rate is given as

\[ C_Q = \frac{\pi}{32 Fr_N^4 \sigma_c} \left( \frac{V_{VORTEX}}{V_\infty} \right) \left( \frac{D_N}{D_c} \right)^2 \left( \frac{L_c}{D_c} \right)^2 \approx \frac{\pi A^2 c_p^2}{32 kFr_N^4 \sigma_c^4} \left( \frac{V_{VORTEX}}{V_\infty} \right), \] 7.2

where the empirical parameters \( A \) and \( k \) are a result of the approximate relations in Eq. 1.18, and \( V_{VORTEX} \) is the exit velocity in the vortex tubes. The value of \( V_{VORTEX} \) is normally assumed equivalent to the free-stream velocity [92], a feature enforced by cavity boundary layers. It was mentioned by Campbell and Hilborne [92] that differences between experiments and theory are a direct result of errors in the assumed value of \( V_{VORTEX} \). In this assumption, when the predicted \( C_Q \) relative to the experiments is high, the modeled velocity in the vortex tubes is also too high. The contrary is also true.

Now consider the two theories with insight based on the computed flow fields. It seems plausible to rewrite Eq. 7.1 with a correction for the separated boundary layer air that is recovered before exiting the cavity. This modification is given as

\[ C_Q = (1-k_{REC}) \left( 1+\frac{\sigma_c}{\sigma_c} \right) \frac{1}{\sigma_c} \left[ \frac{1}{\sigma_c} \ln \left( \frac{1}{\sigma_c} \right) \right] \rightarrow k_{REC} = 1-\frac{C_Q}{C_{Q,BL}}, \] 7.3

where \( k_{REC} \) is the fraction of boundary layer air recovered that assists to effectively reduce the required \( C_Q \). Regarding the physical meaning of \( k_{REC} \), a value of zero corresponds to cavities where all of the cavity boundary layer air is entrained from the cavity. When \( k_{REC} \) is between zero and one, a portion of the cavity boundary layer air is recovered back into the cavity. Negative \( k_{REC} \) values indicate that the total ventilation rate
is higher than that entrained by the boundary layers and an additional mechanism is occurring. When \( k_{REC} \) is very low, the cavity-boundary-layer air becomes a trivial component of the entrained air from the cavity. This situation arises when the cavity is highly ventilated or for high-Reynolds-number flows where the boundary-layer displacement thickness becomes small with respect to the geometry.

Eqs. 7.1-7.2 are plotted in Fig. 7-7(a) for the experiments of a hysteretic, twin-vortex to toroidal cavity attached to a disk cavitator at \( Fr_n=16 \) [92]. Note that \( V_{VORTEX} \) in Eq. 7.2 is adjusted to fit the experimental data using a behavior of \( 1.436 \times 10^{-7} \sigma_c^{-5.07} \). Eq. 7.1 compares well to the experiments for the reentrant cavity and Eq. 7.2 is in agreement with the twin-vortex cavity. These results display a general agreement of the theories for this specific geometry and configuration.

Several plots of the modeled axial velocity in the vortex and the recovery factor, \( k_{REC} \), are provided in Fig. 7-7 (b). The \( k_{REC} \) values are based on the \( Q \)-decreasing experiments plotted in Fig. 7-7 (a). Note that in the toroidal-vortex closure regime, at the
high $\sigma_c$ values, $k_{REC}$ remains near zero indicating that the air is fully entrained through boundary layer mechanisms. In the hysteretic regime, a nearly constant $k_{REC}$ value of 0.53 is observed. The constant $k_{REC}$ behavior indicates that the recovered portion of air in the boundary layer is constant throughout hysteresis. This recovered air is likely the enabling feature that allows a cavity to sustain a larger cavity at a lower ventilation rate, as indicated by the large $k_{REC}$ values. Although data was not collected over a large enough range, it appears that $k_{REC}$ approaches zero quite close to where the theories merge.

The remaining plots in Fig. 7-7 (b) are representations of the axial velocities through the vortex core; that is, the core size based on the cavity pressure and buoyant loads [92]. The axial velocity in the vortex, $V_{VORTEX}$, is based on a fit of the values required for Eq. 7.2 to match the experiments, i.e. $1.436 \times 10^7 \sigma_c^{-5.07}$. The behavior that fits the experiments is quite different from unity, which is the original model assumed [92]. It is useful to break up the axial velocity through the vortex tubes into their potential contributions. The component from shear-layer entrainment, $V_{BL}$, is based on the axial-volumetric-flow rate of air within the boundary layers and the vortex-core area. It is reasonable to assume that cavity boundary layers encapsulate the entire process of air entrainment up to the point where the $V_{BL}$ and $V_{VORTEX}$ curves intersect. At $C_Q$ values higher than this intersection, a different air entrainment mechanism must exist. It is likely that the additional air entrainment is caused by a pressure-gradient driving the air out of the cavity and into the twin vortices. This air entrained through pressure gradients is referred to as $V_{Dp}$, and can be described by $V_{Dp} = V_{VORTEX} - V_{BL}$. Negative values of this quantity are irrelevant, where the cavity boundary layers supply more air than is evacuated from the cavity. When $V_{Dp} > 0$, the amount of air entrained is higher than the
amount of air within the boundary layers. Additional air is evacuated by inflating the cavity to an elevated pressure such that air is driven out of the cavity.

A similar analysis of $k_{REC}$ can be performed on the supercavitating-body experiments [72] (Case AC3) with respect to the numerical simulation approach. In Fig. 7-1(b), the $k_{REC}$ values are plotted for the experiments, RANS, and DES ($Q$ increasing/decreasing) curves. Note that the experimental values show an effective recovery in the apparent hysteretic region. The values of $k_{REC}$ show a similar behavior as the disk cavitation case in the hysteretic regions; however, these $k_{REC}$ values do not remain constant within the region of hysteresis. Such differences may be expected as the cause of the hysteretic behavior in this case is completely different. In this case a geometry induced hysteresis occurs, as compared to the disk-cavitator having closure-type induced hysteresis. Contrary to the experiments, the RANS-based simulations display very little air recovery, which agrees well with the $Q$-increasing DES curve. The $Q$-decreasing DES curve displays a similar $k_{REC}$ behavior as the experiment. Recall that the URANS cases also captured a similar behavior as the DES case. By comparing the URANS and RANS predictions, the capture of this air recovery may relate to the unsteady integration of the simulation. The air recovery also appears to be a possible indicator of hysteretic regions in the cavity behavior.

Based on these investigations, it seems reasonable to propose a new form of an air entrainment model. A more inclusive model would include cavity-shear layer effects and when the twin-vortices become large enough, introduce a pressure-gradient based model. It is proposed that such a model could be formed as
The model needs empirical calibrations based on the $Re_N$ and $Fr_N$. In particular, the modeled quantities include $k_{REC} (\sigma_c, Re)$, $V_{Dp}/V_\infty (\sigma_c, Re)$, and $k_Q (Re)$. As $Re \to \infty$, it is expected that Reynolds number independence is eventually achieved. Approximating these relations could be performed through experimental or computational data points. Unfortunately, not enough data is available in the open literature to provide such an empirical model. Based on this single case, a model can be determined empirically. For example, a reasonable fit of the pressure gradient driven outflow could be empirically modeled as

$$C_Q = (1 - k_{REC}) k_Q \frac{(1 + \sigma)}{\sigma} \sqrt{\frac{1}{\sigma} \ln \frac{1}{\sigma} + \frac{\pi A^3 c_p^2}{32 k_{Fr_N}^4 \sigma^4}} \max \left( \frac{V_{Dp}}{V_\infty}, 0 \right).$$  \hspace{1cm} 7.4$$

The theoretical minimum cavity pressure, $\sigma_{c, \text{min}} = 1.34 \sqrt{C_{D,0} Fr^4}$, is the most obvious reference for these empirical fits. This naturally enables the exponential in Eq. 7.5 to asymptote to enforce the correct minimal $\sigma_c$ behavior. A sample case is given for disk cavitator experiments [92] in Fig. 7-8. Note that the actual model suggested by Campbell and Hilborne [92] is used in this comparison. The proposed model improves the predicted behavior near the hysteresis region. This should be expected considering that the empirical behavior is established for this specific case. Thus, it is clear that additional data points need to be evaluated. The author feels that the general form of the model is

$$\max \left( \frac{V_{Dp}}{V_\infty}, 0 \right) \approx \frac{26.1 e^{273 (\sigma_c - \sigma_{c, \text{min}})}}{(\sigma_c - \sigma_{c, \text{min}})^{0.2}},$$  \hspace{1cm} 7.5$$

and the recovery factor as

$$k_{REC} \approx \max (-5400 (\sigma_c - \sigma_{c, \text{min}})^2 + 302 (\sigma_c - \sigma_{c, \text{min}}) - 3.56, 0).$$  \hspace{1cm} 7.6$$
more inclusive as it accounts for viscous effects applied for a twin-vortex closing cavity, but the modeled parameters need a deeper evaluation.

7.1.3 Summary of Supercavitating Flow Insights

Several simulation capabilities and insights of supercavitating flows have been observed in this work. First is the ability to predict hysteretic behavior in these cavitating-fluid flows is shown. Such a capability displays strong potential in the solutions method. Next, through numerical-solution insight, the basis of models suggesting that cavity-shear layers affect the cavity-air entrainment was observed in these simulations. Such insight allowed the development of novel forms of an air-entrainment model that display potential to be valid throughout the entire range $C_Q-\sigma_c$ curve for a cavity. This is a missing component in twin-vortex cavity models of air entrainment. These insights should enable improved modeling capabilities for supercavity theory.

Figure 7-8: Proposed form of modeling $C_Q-\sigma_c$ relation.
7.2 Interactions of Cavitation with a Free Surface

Cavitating bodies and lifting surfaces are investigated when operating near a free surface. As both natural and artificial cavitation types are of interest, both are examined. These studies initiate with rather simple geometries, then progress into simulations representative of full-scale submerged vehicles.

7.2.1 Flat-Plate Predictions Compared to Boundary Element Theory

Predictions with a CFD approach are compared to those from a boundary element approach [93] for the supercavitating flow of a flat plate near a free surface. In the boundary element approach, the $\sigma_c$ value is specified, while a CFD approach, similar to experiments, requires the specification of the ventilation rate. This feature causes the CFD process to become iterative; thus, the CFD $\sigma_c$ value is only close to the boundary-element prediction $\sigma_c$ value of 0.05. Note that in the CFD solution a turbulence model is not used; this is justified by the expected laminar boundary layers and the sharp edge initiating separation.

Figure 7-9 shows predictions for a flat plate positioned 1 $c$ beneath the undisturbed free surface, with reference to the trailing edge (T.E.). Comparisons between predictions from a CFD and boundary element method [93] are made. The CFD cavity shape and resulting free-surface fields are not in good agreement with the theory. One major deviation in the CFD has a simulated, rather than modeled, closure behavior. In Fig. 7-9 (a), the CFD predictions show a smaller cavity than those predicted with the
boundary element method, with the CFD having a slightly lower cavitation number. The cavity length, from Faltinsen and Semenov [93], is indicated for the CFD cavity pressure with the vertical-dashed line. For the $Fr_c=5$ case, in Fig. 7-9 (b), the CFD predicts a larger cavity with a lower $\sigma_c$, which is the expected trend. Note that the differences in the cavity and free-surface predictions correlate quite well. Finally, in Fig. 7-9 (c), the ventilated flat plate is examined at $Fr_c=\infty$. The CFD cavity is smaller than that of the boundary element method; this feature is expected as the CFD cavity has a larger $\sigma_c$ value. Again, the discrepancies in the cavity and free-surface predictions correlate quite well. In general, the increased Froude number cases show comparable predictions based on extrapolating the cavitation number. However, in the lower Froude number case there is an unaccountable discrepancy.

Figure 7-9: Cavity and free-surface predictions occurring near a ventilated flat plate. The flat plate is positioned 1.0 c beneath the undisturbed free surface. The CFD predictions are compared to boundary element method predictions from Faltinsen and Semenov [93]. The upper line indicates the free surface, whereas the oval-shaped line represents the cavity interface. The dashed vertical line is a corrected cavity length from the theory [93] based on the CFD computed $\sigma_c$. 

(a) $Fr_c=2.0$ ($\sigma_c=0.0479$) (b) $Fr_c=5.0$ ($\sigma_c=0.043$) (c) $Fr_c=\infty$ ($\sigma_c=0.0547$)
7.2.2 Three-Dimensional Predictions

Noncavitating Free-Surface Interactions

The hydrofoil geometry used in the oscillating, cavitating fin experiments [77] is investigated under similar conditions performed in the previously examined submerged hydrofoil experiments [78]. The conditions of this case correspond to the following: $Fr_c = 0.57$, $Re_c = 4 \times 10^5$, $\alpha = 5^\circ$, and the hydrofoil is submerged at $d/c = 0.783$ at the quarter chord. The geometry is symmetric, has a fifteen percent maximum thickness, and an aspect ratio of 4.0. One-half of the span is actually modeled where a symmetry plane is placed at the base of the wing. All cases use a grid having $2.02 \times 10^6$ cells, which is rather high because of the intent to resolve an unsteady vaporous cavity on the foil.

In Fig. 7-10, the effect of the free surface on the wake of the hydrofoil is displayed. Here, the net velocity is downwards due to the induced velocity from the trailing tip-vortex system, which is a direct result of the lift produced from the hydrofoil. Within the induced flow from this vortex, additional velocity components resulting from the free-surface deflections are observed in the y-direction, or vertical, velocity component. Note that this feature is not observed in the z-component of the velocity. Furthermore, the wake displays a fluctuating pressure and vorticity resulting from the free surface.
In Fig. 7-11, are the predicted free-surface elevations with an inviscid and a far-field boundary placed 3.0c from the wing tip. The inviscid wall is representative of a tow-tank experiment, while the far-field boundary condition is representative of an open water configuration. In comparison to the two-dimensional NACA 0012 hydrofoil experiments, the predicted wave elevations near the foil are slightly lower and no breaking wave is apparent. [78] Although the foils are different, theoretically the lift is close. The major feature that differs is the pressure distribution, in particular the lower suction peak of this hydrofoil implies a lesser wave-driving mechanism. Regardless, the surface elevations are close to the experiment and the author is confident in these predictions. Comparing the two cases, the general profiles are identical until the waves strike the side-boundary conditions. The inviscid wall reflects the wave and appears to strengthen and bend the subsequent wave. For the far-field boundary condition the waves appears to dampen artificially. This is expected to be a boundary condition issue, where the pressure is fixed.
based on the free-stream conditions and these waves are not properly considered. Regardless, they remain of little significance to the predictions near the hydrofoil.

**Naturally Cavitating Wing Interacting with a Free Surface**

A cavitating wing submerged beneath a free surface is investigated, where the geometry and mesh from the previous submerged wing cases is extended to a natural cavitation state. The results are presented in Fig. 7-12. Here, a $Fr_c$ of 9.23 is used so that the surface pressure remains above the vapor pressure for $\sigma_v=0.5$ (referenced from the surface pressure). In Fig. 7-12 (a), the predicted cavity is shown in blue and is compared to the noncavitating result. Although the cavity is rather small, it is unsteady and possesses a reentrant jet on the inboard regions of the back half of the hydrofoil as presented in Fig. 7-12 (c). The predicted free-surface elevations for both cases are displayed in Fig. 7-12 (b). Here, it can be observed that the initial wave of the cavitating case has only a slightly different amplitude. Considering the trough that follows the initial
wave, in the cavitating case the transition is much longer than for the noncavitating case. This indicates that the cavitating case has a much longer wavelength than the noncavitating wing. Note that the entire domain is shown in this plot and that, for these high $Fr_c$ cases, the required domain size, hence number of computational cells, increases rapidly for increased $Fr_c$. Based on these results, the cavity actually reduces the induced wave field. This is likely an effect of cavitation effectively limiting the pressure minimum on the foil. It appears as if the pressure minimum, rather than the increased effective thickness, is most important in driving this wave field. Future analyses would recommend investigating this same case with a larger computational domain.
Figure 7-12: Side by side comparison of the free surface prediction for a cavitating ($\sigma=0.5$) and noncavitating control fin, submerged $0.783c$ below the surface, at $Fr_c = 9.23$. (a) The cavitating case is displayed with the blue isosurface representing the cavity, and is compared to the case with no cavity. (b) Comparisons of the predicted free-surface elevations. (c) The predicted reentrant jet forming in the cavitating simulations.
7.2.3 Ventilated Cavitation

*Lifting Surfaces*

Scherer and Auslaender [12] presented measurements obtained from the flow over a supercavitating hydrofoil submerged beneath a free surface. Their results ranged over angles of attack from 3 to 15 degrees, vaporous cavitation numbers, $\sigma_v$, from 0.06 to 2.34, $Re_c$ from $1.16 \times 10^6$ to $2.01 \times 10^6$, $Fr_c$ from 8.19 to 14.2, and submergence depths, $d/c$, of 0.5 and 1. The geometry is a cambered, wedge-shaped hydrofoil ventilated at the base of the supporting strut. In the experiment, the cavity was filled with gas at a pressure just slightly less than atmospheric, where the cavitation number was measured to be less than 0.04. Contrary to the experiments, ventilation ports, rather than entrained air from the free surface, are used to ventilate the cavity. This implies the need for an iterative strategy to achieve the same cavitation number, a process yet to be performed. The specific cases studied include a span of $1.5c$, $d/c=0.5$, $\alpha=10^\circ$, $Re_c=1.16 \times 10^6$, $Fr_c=8.19$, and a vaporous cavitation number of 0.25. Two cases are considered, a vaporous case and a ventilated case similar to the experiments. In the presented ventilated case only two phases were modeled (liquid-water and air), as the cavity has a pressure greater than the vapor pressure, little vapor should be present and is neglected.

Figure 7-13 contains selected results from computations of the predicted flow over this supercavitating hydrofoil. In Fig. 7-13 (a), the $\alpha=0.5$ isosurface displays the predicted cavity shapes. Note that the cavity may visually appear closer to the $\alpha=0.1$ isosurface, which is shown in Fig. 7-13 (d). In Fig. 7-13 (b), the cavity is colored by the height. The vaporous cavitation case is nearer to the free surface than the ventilated case.
In addition, the vaporous case contains an unsteady closure region, whereas the ventilated case is rather steady. The predicted free-surface elevations due to both cavities are displayed in Fig. 7-13 (c) and (d). Both cases display a complex wave field having three-dimensional waveforms. The free surfaces from both cases display dominate initial swells that are initiated due to the foil and cavity. This swell is larger for the vaporous cavitation case, which is a direct result of lower pressure in the cavity. Both swells are smooth and transition into quite different depth troughs. The major difference in the length of the trough, which is a result effectively extending the body length with the larger cavity observed in the ventilated case. Further downstream, a secondary wave appears in the center of the wake. Note that the tip vortices elevate toward the free surface, a result of buoyant loads on the tip vortex. The primary waves are coincident with the elevating tip vortex structures that follow the outer edge of the cavity. Furthermore, as displayed in the ventilated case in Fig. 7-13 (d), the free surface and cavity both have a similar downwash associated with the lifting surface. In general, the cavity and free surface display strong interactions.
Figure 7-13: Comparison of vaporous and ventilated cavitation predictions from a supercavitating hydrofoil. The conditions correspond to $Fr_c = 8.23$, $\alpha = 10^\circ$, $Re_c = 1.4 \times 10^6$. The ventilated case has a $\sigma_c = 0.04$ whereas the vaporous case uses $\sigma_v = 0.25$. In parts (a) and (b) the $\alpha = 0.5$ isosurface displays the cavity shape.
Axisymmetric Bodies of Revolution

A demonstrative case comparing the induced wave field due to large-scale axisymmetric objects near a free surface is presented. The flow over a 4:1 spheroid that is submerged one-half body length beneath the surface is compared to the flow over a body shape that is deliberately made to cavitate. The body with the cavitating head form has a flat nose of diameter one tenth of its axial length. The rest of the body is made of two spheroidal sections. From the nose to the equator, a truncated spheroid smoothly mates with the 4:1 hemispheroidal aft end. Both objects are run at the same $Re_L=2.9\times10^8$ and $Fr_L=3.12$. The cavitating body is run at $\sigma_c=0.318$ (based on the centerline free stream depth pressure).

The wave field induced by the two body shapes is compared in Fig. 7-14. Although the initial swells generated by the bodies are similar in amplitude and shape, the similarities end there. The wave field induced by the blunt-nosed body appears broader and, based on the limited wave field displayed in the solution, has a longer wavelength. It is clear that for long range wave-field analyses, some more efficient method of capturing the far field would be useful. Such an approach might be that of Iafrati and Campana [51].
In Fig. 7-15, the surface pressure on the two axisymmetric bodies and the near field free surface are displayed. In addition, the inherently unsteady partial cavity is illustrated on the blunt-head form with an isosurface of vapor volume fraction equal to 0.5. Note that the shallowest surface at the widest point of the body displays a cavity, while on the lower part this is not apparent.
In Fig. 7-16 is a comparison of the computed flows around a fully wetted and ventilated supercavitating body, both having a 4:1 aspect ratio. This ventilated case is an unsteady calculation. For the supercavitation case, the dimensionless ventilation rate is $C_Q = 1.05$. The vehicle uses a disk cavitator with two concentric gas ports, angled to direct the air against the free-stream water, at 45 degrees to the horizontal. The aft half of the vehicle is a hemispheroid. The cavity about the vehicle is illustrated with a gray isosurface of liquid volume fraction equal to 0.5. The free surface is also illustrated with an isosurface of the liquid volume fraction equal to 0.5 and is colored by the height above the nominal waterline. Both bodies are submerged one-half body length beneath the surface at their centerline, and are running straight ahead at zero angle of attack. The conditions correspond to $Re_L = 2.9 \times 10^8$ and $Fr_L = 3.12$. The free surface is colored by height, in body lengths, above the nominal waterline. The bodies and cavity are visible through the translucent free surface.
Comparisons of the fully wetted and supercavitating predictions are based on the observed predictions in Fig. 7-16. In the absence of a downstream cavity, in Fig. 7-16 (a), the body influence on the surface flow results in a small, nearly circular, upward deflection pattern directly above the spheroid. Downstream of the spheroid there is an interesting shallow depression in the wake. This eventually dissipates into two divergent depression streams. In Fig. 7-16 (b), the predicted wave field is significantly different from that of the fully wetted spheroid. The flow about the buoyant twin-vortex cavity creates a significant rise in the free surface that persists far into the wake of the vehicle.

A representation of the supercavitating case is displayed in Fig. 7-17. This rise is surrounded by a mild depression. Fig. 7-17 (a) illustrates the well-resolved twin-vortex cavity closure in the wake of the vehicle. Here selected (instantaneous) streamlines that originate from the ventilation ports are highlighted. Within the cavity, the streamlines follow a circuitous route, which is representative of the turbulent flow expected within the supercavity. Also, in Fig. 7-17 (a), contours at three axial locations have been selected...
downstream of the vehicle and are colored by the gas volume fraction. These illustrate the transport and separation of the vortices. Note that the turbulent simulations presented do not assume a symmetry plane on the flow field around this supercavitating vehicle. In the view given in Fig. 7-17 (b), the twin vortices are highlighted by the isosurface of liquid volume fraction equal to 0.5. The vortex structures exhibit physical meandering as they tail away from the vehicle. Finally they terminate (as resolved by the selected isosurface) beneath the maximum surface rise downstream in the vehicle wake. Note that the waterline surface height is colored by the elevation from the undisturbed state. In Fig. 7-17 (c), is a view representing the flow field near the vehicle. In a plane in the wake, vectors are plotted displaying the effect of the vortical structures on the surrounding flow and the free surface. Note the induced impinging jet acting on the free surface, which is the predicted feature that appears to be causing the elevated free surface in the vehicle wake.
7.3 Investigations of Oscillating, Multiphase Lifting Surfaces

Several oscillating hydrofoils are evaluated and studied in various multiphase flow configurations. These studies include hydrofoils operating near a free surface and when cavitating. The goal is to improve the understanding how these hydrofoils function in such configurations.

7.3.1 Oscillating a Hydrofoil Submerged Beneath a Free Surface

Reconsider the oscillating hydrofoil simulations used in the validation in Section 5.18.42.1, when submerged beneath a free surface. Comparisons of the lift and drag are
provided in Fig. 4-14, when the hydrofoil is pitched from -5°<\(\alpha<25°\), for various submergence depths. These results indicate a moderate effect on the hydrofoil loads with a varied depth of submergence (from \(d/c=1.29\) to \(d/c=0.783\)). Note the nearly constant reduction of the lift throughout the linear-lift region of the pitch cycle. The stall-angle of attack for the dynamic hydrofoil is delayed with decreased submergence depths. Considering the contrary pitch-direction case, that is ranging from -25°<\(\alpha<5°\), a slightly different trend is observed. The linear-lift region remains the same, as expected. Both the stall angle and maximum lift decrease compared to those for the positive-pitch case. The hydrofoil also appears to have a difficult time recovering from stall. This is observed in the lift behavior near \(\alpha=-5°\), where the prediction varies significantly from the linear-lift behavior resulting from a lag in the recovered attachment of the separated flow region. Future research should examine this behavior in additional detail.

7.3.2 Oscillating Fin in a Water Tunnel

Evaluations of oscillating hydrofoils that are cavitating are made based on experiments presented in Kinzel et al. [77] (Case VC3). Although the experiments use a three-dimensional fin, two-dimensional simulations are also performed to more efficiently investigate important parameters of the problem. Then, the solutions are extend to three-dimensional simulations.
7.3.2.1 Effect of Pitch Rate

The effect of pitch rate is examined by scaling the time in the simulations by a factor, \( f_t \), that reduces the physical time modeled in the computation. This scaling is intended to reduce the number of cavitation shedding events simulated, which occur at a much higher frequency than the wing is pitched and still need to be resolved. In terms of a reduced frequency, \( k \), the experiments are well into the quasisteady realm. In this case, \( k = 0.0025 \), whereas the typical quasisteady oscillation rate for single phase is \( k = 0.05 \). Using the same pitch cycle, at a scaled time, the pitch rate can be increased to a point before unsteady effects are important. As this pitch motion is not based on harmonic motions, some deviation is expected at the events where the motion is most rapid. Where cavitation occurs, the pitch rate is much milder in the cycle.

Examined Pitch Rate for Noncavitating Oscillations

In Fig. 7-18, are comparisons of the scaling for single-phase simulations. The appropriate \( f_t \) is varied to scale the oscillation cycle to give \( k = 0.025, 0.05, \) and \( 0.5 \), corresponding to \( f_t = 0.1, 0.05, \) and \( 0.01 \), respectively. Note that \( k = 0.05 \) corresponds to the unsteady-quasisteady limit for harmonic motions. The difference in the predicted lift and drag is small between the cases where \( f_t = 0.05 \) and \( 0.1 \), thus, the simulations remain quasi-steady. The unsteady effects become important when \( f_t \) is reduced even further, to values of \( 0.01 \). As expected, the influence of the pitch rate and shed vorticity is small enough to assume that the aerodynamics behave quasisteady while \( f_t > 0.05 \).
Examined Pitch Rate for Cavitating Oscillations

A similar study of scaling the pitch cycle in time is performed when the hydrofoil is cavitating. In this case, the cavitation number is reduced to $\sigma_v = 0.6$ and the oscillation cycle now exceeds the static-stall angle of attack, thus, simulates a cavitating-based dynamic stall. Varied values of $f_t$ are used to scale the oscillation cycle to give $k=0.0025$, 0.005, 0.025, and 0.05, corresponding to $f_t= 1.0, 0.5, 0.1$, and 0.05, respectively. The lift and drag predictions are plotted in Fig. 7-19, where smoothing of the $f_t= 1.0$, 0.5, and 0.1 cases is used for clarity. Note that the loitering period between the two cycles was removed to shorten the simulation time. In general, the lift responses are much more sensitive to the pitch response than was observed in the single-phase cases. The most noticeable difference is in the first cavitation event, which appears to be a relatively slow process and dominates the initial stall behavior. Using a faster pitch rate, the maximum lift increases to values comparable to the single-phase lift. At a decreased frequency, the lift displays an initial drops at the some point ($t^* \approx 0.225$). Following this first cavity cycle, the behavior of the mean lift response is consistent at all reduced frequencies until
the point where the foil is recovering from the cavity (roughly $t^*=0.75$). It is clear that recovery from the cavitation is also dominated by the pitch rate, and is likely associated with a finite time required for the cavity to advect away from the hydrofoil. At the lower frequencies, the mean lift tends to reflect the pitch cycle. At increased frequencies, the lift is lesser in the nose pitching down phase, which is perhaps amplifying the cavity. At some point, a minimal lift is achieved and a rather slow recovery is observed. These results indicate stronger sensitivities to the pitch rate than single-phase cases, and the quasisteady behavior is only observed when $k<0.005$ ($f_t=0.5$).

The flow fields of the initial cavity shedding cycle are displayed in Fig. 7-20 below. These plots can be referenced to the times with the vertical-dashed lines in the load predictions plotted in Fig. 7-19. The contour plots display the vapor-volume fraction (in grey) highlighting the cavity shape. A corresponding surface pressure plot is also included to give insight on the nature of lift generation. In the figure, it is apparent that

![Figure 7-19: Effect of pitch frequency on the predicted loads of a two-dimensional simulation. Note that the $f_t=1.0$, 0.5, and 0.1 load predictions are smoothed for clarity; the actual fluctuations are near those of the $f_t=0.05$ case. The vertical-dashed lines correspond to the flow-field visualizations in Fig. 7-20.](image-url)
the cavity shape is quite different amongst all pitch rates. Figure 7-20 (d) displays the highest pitch rate case, which also has the highest lift in the initial cycle. The distinguished feature of the high-sustained lift in this case is the stable cavity that maintains the vapor pressure on the suction surface of the hydrofoil. The moment the cavity destabilizes, as displayed at $t^*=0.40$, the lift is lost. The major feature of the cases that are not achieving a high lift is the collapsing of the cavity on portions of the suction surface. These collapsing events correspond to high-local pressures from the jet of water that follows the collapse. Lastly, it is important to note that the high-lift event is not unique to the high-pitch rate case. Similar events are observed in the low-frequency cases, but are removed in the averaging. This indicates that this event corresponds to the advection times. When the pitch cycle approaches the cavity shedding frequency, as observed in the $f_*=0.05$ and 0.1 cases, the foils dynamically takes advantage of additional lift associated with the shed cavity.
Three-Dimensional Simulations

Now consider the three-dimensional simulations, which are more representative of the experiments. The three-dimensional cases are computed using a $f_t=0.05$. Based on the previous two-dimensional simulations, the noncavitating simulations should be

Figure 7-20: Comparison of the predicted flow field during the first cavity cycle for various pitch rates. Each row is at a similar normalized time, $t^*$, and $\alpha$, which can be referenced to the loads in Fig. 7-19. The scaling factors are indicated at the bottom of each column. The contour plots display the vapor-volume fraction (grey indicates the amount of vapor). The line plot displays the corresponding foil-surface $-c_p$ values.

7.3.2.2 Three-Dimensional Simulations

Now consider the three-dimensional simulations, which are more representative of the experiments. The three-dimensional cases are computed using a $f_t=0.05$. Based on
unaffected by the time scaling, however, in the cavitating simulations, differences from
the experiments are expected that should be similar to the differences in the $f_t=0.05$ and
$f_t=1.0$ cases of Fig. 7-19.

In Fig. 7-21, are the predicted lift and drag of the two- and three-dimensional
cases with $\sigma_v=0.6$ and 4.0. The lift can be compared to the experimental measurements.
Note that the two-dimensional predictions of lift use three-dimensional extensions based
on lifting-line theory and the $\sigma_v=4.0$ case applies a correction based on the effects of
modeling transitional flow. The two- and three-dimensional lift predictions display some
deviations when cavitating ($\sigma_v=0.6$). This is expected as lifting-line theory is a linearized
theory that does not consider effects from stall. When $\alpha$ is decreasing, both of the two-
and three-dimensional results display the behavior previously observed in the studies in
Fig. 7-19.

Figure 7-21: Various prediction methods of the finite wing hydrodynamics.
Further evaluations of the predictions compared to the experiments have been made between the computed cavity shape and the shape observed experimentally. These can be referenced in Figs. 6-10 and 6-11. In general, all three-dimensional predictions remain in good agreement with the observed cavity shapes from the experiments. Note that the video was not performed at a high enough frame rate, thus, does not necessarily correspond to the same angle of attack. Regardless, after stall, the cavity dynamics should be insensitive to the pitch angle and the cavity patterns should be reproducible, which is shown in the predicted cavity shape.

Consider the predicted flow features between the two- and three-dimensional flow fields displayed in Fig. 7-22. In this case, the two-dimensional flow field represents the three-dimensional flow field quite well in the general cavity development, stability, and general size. Obviously, some of the details are not encapsulated by the two-dimensional case, which includes the tip vortex and surrounding flow, the three-dimensional shape of the cavity, and three-dimensional reentrant jet in Figs. 7-22 (d-e). The surface pressure distributions of the three-dimensional case are consistent with the two-dimensional predictions, especially away from the tip. The most deviant time is in Fig. 7-22 (e), where the reentrant jet forms. The three-dimensional behavior of the reentrant jet enables the hydrofoil to sustain the higher lift values than the two-dimensional case does. This is due to effectively reducing the size of the jet with respect to the planform area. In the two-dimensional case, the jet extends along the entire span, whereas for the three-dimensional case, it actually acts on about quarter of the span. Thus, the cavity is capable of maintaining lower pressures on the suction surface for the three-dimensional case.
Figure 7-22: Comparison of two- and three-dimensional predictions at $t^*=0.05$. In each subfigure, the upper plot is a three-dimensional solution representation and the lower is from a two-dimensional prediction. The 3D case displays the surface streamlines (in red), vapor-volume fraction isosurfaces at 0.5 (pink) and 0.1 (white), and has solid surfaces colored by pressure. The surface pressures at three spanwise stations are also provided (red => $z/c=0.25$, yellow => $z/c=1.0$, and green => $z/c=1.8$). As before, the two-dimensional predictions display the vapor-volume fraction in grey and the surface pressure distribution.
The CFD results are also useful in that they yield the entire flow field at each computational time. Additional investigations of these results are displayed in the predicted reentrant jet in Fig. 7-23 below. In the plot, the cavity is displayed by the vectors at the various spanwise locations, and the blue regions on the wing surface. The initially stable cavity shows a recirculation region forming at the aft end of the cavity, in Fig. 7-23 (a). In Fig. 7-23 (b), the cavity grows and remains stable; however, water continues to fill the rear portion of the cavity at the foil surface. At a later time, in Fig. 7-23 (c), the accumulating water converges into a jet that penetrates into the center of the cavity. At this point, water is entrains from the pressure surface of the wing and forms a cavitating separation bubble at the center of the trailing edge. In Fig. 7-23 (d), the reentrant jet reaches the wing leading edge and the waters jet accumulates at initiation point of the cavity. This jet eventually appears to destabilize the cavity. Then, as shown in Fig. 7-23 (e), the cavity vapor and water violently mix. In this mixing, multiple spanwise collapsing events occur, which are indicated by the observed impinging jets where the surface streamlines merge. Immediately following this mixing, the cavity violently breaks down due to the resulting increase in cavity pressure. This pressure increase causes the cavity to collapse, which entrains a jet of water that impinges on the wing surface. This jet is shown in Fig. 7-23 (f), which is due to the abrupt density changes from the condensing vapor. These events describe the predicted initial cavity cycle, which is expected to repeat itself in a static or slowly pitched case.
Figure 7-23: Predicted events of a reentrant jet and cavity collapse. The surfaces are colored by the liquid-volume fraction. Vectors of the vaporous regions are plotted at various spans along the wing. The black lines are surface-streamlines.
7.3.3 Summary of Oscillating Multiphase Lifting Surfaces

Several oscillating lifting surfaces operating in multiphase flows are investigated. The first case is a submerged hydrofoil operating near a free surface. In this case, there are effects of the free-surface interacting with the hydrofoil are clearly observed. These effects appear to relate to the submergence depth and the lifting direction with respect to the free surface. Secondly, cavitating cases were considered. Two-dimensional analyses of the pitch rate displayed significantly different quasisteady-unsteady boundaries than in singlephase flows. This is a result of the highly unsteady cavitation events. Furthermore, extensions to three-dimensional wings were also examined. Comparisons of two-dimensional and three-dimensional analyses support the usage of two-dimensional data for three-dimensional studies, as done in lifting-line theory. Furthermore, the destabilization of the cavity was also observed. The solutions display that the reentrant jet forms and entrains water from both the suction and pressure surface, then impacts the cavity creating the unstable cavity behavior that eventually leads to the loss of lift.

7.4 Summary of Investigations

The CFD simulations were used to investigate the physics of complex, cavitating flows. First, hysteresis caused by geometric features of supercavitating body was investigated. These studies led to the examination of air entrainment mechanisms from reentrant and twin-vortex-type cavities. Based on the observed flow-field features, a more inclusive modeling approach to air entrainment was proposed. Next, cavitating objects near a free surface were evaluated and the effect, compared to noncavitating cases and
different cavitation types, displays a strong impact on the resulting wave field from the body. The effects suggest that the minimum pressure, typically lesser in cavitation problems, and the cavity size, correlate to the size of the waves. Finally, oscillating foils and wings were investigated in multiphase configurations. Effects of the hydrofoil oscillations in dynamic stall and near a free surface were evaluated, the results display a small but relevant effect in the range studied. In addition, cavitating foils that oscillate were investigated. These studies investigated the impact of oscillation frequency and extensions to three-dimensional wings. Furthermore, the process of the formation of the reentrant jet was investigated. In general, the CFD predictions show an abundance of insight into all of the investigated applications.
Chapter 8
Conclusions

In this thesis, the numerical methods, necessary models, and a physical understanding of large-scale cavitating flows were developed. The baseline method of UNCLE-M was extended to include additional simulation capabilities and improve the numerical approach for discrete and turbulent interfaces. Computational-mesh approaches based on overset and adaptive meshes were examined; the approaches show improved computational efficiencies and an extended ability to simulate dynamic-multibody problems. The numerical approach based on the level-set interface-capturing technique is tailored to the physical requirements of cavitating-fluid flow simulations. These additions enable a tool that can effectively simulate the complexities needed in multiphase vehicle design.

Within this work, the CFD code was extensively validated for multiphase flows. The relevant applications include the analysis of multiphase underwater vehicles, cavitating pumps, and cavitating rocket propulsors. The method shows the capability of predicting an extensive range of cavitation types, while remaining valid for free-surface flows. Furthermore, the design of the code enables the evaluation of thermodynamic and turbulent flow assumptions that were useful in understanding their impact to the prediction. Finally, the validated CFD models and methods were used to extract physical insight into features of such complex flows.
8.1 Summary of Research

Numerical Model Improvements:

Several concepts were evaluated to improve the predictive capability of interfacial fluid flows. These concepts include overset and adaptive grids, inviscid flux variations, as well as the level-set method. The overset grid method adequately predicts cavitating flows and is valuable in that it simplifies the extension of traditional structured-meshing methods to handle complex and dynamic simulations. The adaptive method investigated improves the efficiency of the solver by better utilizing the computational cells for free-surface predictions. It is expected that the concept is more useful in its incorporation into more advanced adaptive algorithms. The examinations of various inviscid-flux formulations displayed that the form used, for a shock-capturing interface treatment, is key in preventing an overly diffusing interface. Finally, the signed-distance-based level-set approach for fluid interfaces was analytically transformed into a volume-fraction-based approach. In this form, compressible and mass-transfer formulations easily extend from mass-species conservation formulations. In addition, a more physical reinitialization procedure is developed that retains the capability to predict subgrid-scale interfacial features, an essential feature for cavitating flow predictions. The method is computationally efficient and straightforward to incorporate into a CFD code.

Validation of the Numerical Method:

The general numerical method was thoroughly validated for cavitating and free-surface flows. The validations include standard cases and extend to novel CFD simulations. Ventilated cavitators, submerged hydrofoils, and free-surface-piercing
hydrofoils are some of the traditional validated applications. The application of UNCLE-M was extended and validated for cavitating cryogenic-liquid flows. Novel CFD cases explored include supercavitating hysteresis, cavitating objects beneath a free surface, and cavitating-oscillating lifting surfaces. In general, validation difficulties seem to originate from the modeling assumptions. The nonlinear nature of the examined flows tends to be unforgiving for seemingly valid assumptions. The problematic cases range from neglected geometric features, turbulent flow models, inflow boundary conditions interacting with enclosed bodies, importance of unsteady integration, and the thermodynamic behavior. With an appropriate computational model and modeled physical behavior, the presented numerical schemes enable a successful simulation capability.

**Physical Model Improvements:**

In this work, a rather careful and consistent study of the physical modeling requirements was evaluated. The studies focused on the impact of the modeling approaches for turbulent and compressible fluid flows. In general, eddy-resolving simulations improve the predictive capability for both artificial and natural cavitation. Although the RANS approach displays accurate predictions, cases arise where the model becomes inadequate. In particular, hysteretic cavities and highly dynamic partial cavities were difficult for unsteady RANS methods to predict. Using a DES modeling approach, one that remains sufficient for the high-Reynolds number applications of interest, the issues encountered with RANS are overcome. The correct modeling of the multiphase thermodynamics also shows its importance, especially for compressible-multiphase flows. In particular, using isothermal assumptions improves the predictions. However,
condensation- and evaporation-induced temperature changes display additional effects due to the subsequent liquid-density changes. These effects are known to be important with cavitation in a cryogenic-liquid, but also show importance with cavitation in water, at least in select cases. In general, using more inclusive physical modeling improves the reliability of the predicted solutions.

**Physical Insight into Cavitating Flows:**

The validated models are used to gather physical insight. Several physical concepts were investigated that include the hysteresis of artificially ventilated cavities, cavity-air entrainment, free-surface interactions with cavitating flows, and multiphase oscillating-lifting surfaces. These studies were focused towards fluid-flow insight and design-level model improvements for flows that are difficult to analyze experimentally. First, supercavitating hysteresis around an enclosed body is studied. These studies extend to a detailed investigation of air entrainment mechanisms and approaches to improve the current models. Next, the effects of cavitating objects near a free surface are evaluated, which significantly impacts the resulting wave field. Finally, oscillating lifting surfaces are investigated in several multiphase configurations. The largest impact observed was for cavitating hydrofoils that oscillate. In such cavitating cases, quasisteady hydrodynamics are not achieved until pitching at reduced frequency below 0.005, as compared to single-phase cases that are quasisteady for values less than 0.05. Furthermore, insight into a three-dimensional reentrant jet and the subsequent cavity collapse events are presented.
8.2 Future Work

*Improved Adaptive Mesh Algorithms:*

Several aspects relevant to adaptive-mesh algorithms were evaluated, but not fully addressed. A surface-tracking volume-based method shows strong potential to improve the computational efficiency and the quality of the results. Furthermore, mesh refinement studies display that the predicted wave dynamics relates to interface sharpness as well as wave-induced motions beneath the free surface. A lack of resolution of the wave-induced motions artificially dampens the predicted wave motion. It is suggested that future adaptive-mesh algorithms would be greatly improved by including a topologically adapting mesh that conforms to both the interface and the underlying wave-induced motions.

*Advancement of Developed Level-Set Method:*

The developed level-set method was tested for artificially cavitating applications and high-speed shock-tube problems. However, many applications and flows exist where the method should be considered. The method shows potential as a more efficient, easy-to-implement, and an equally effective level-set approach that does not artificially sharpen the subgrid-scale mixtures. Furthermore, the local species mass losses can be determined locally and a conservative level-set approach can be achieved. Such additions would be invaluable to the level-set approach for interfacial-fluid flow applications.

*Improved Convergence Methods:*

One of the issues that remain in these cavitating fluid flows is the long times associated with the transient part of the solution. The reduction of the integration time
required in these transient solutions is, perhaps, where the efficiency can be most improved. The issue arises from the long time scales associated with cavitating-fluid flows, a result of influences from the gas propagating with the flow particles. The implication is that the error propagations strongly correlate to the local, rather than acoustic, speeds. Furthermore, the observed strong influence in the thermodynamic properties tends to limit the solution method to small time-step sizes. The general implication is that more accurate, and perhaps efficient, routes may use explicit-type time integration methods. These are more efficient per time step, and can remain stable for the small time-step sizes needed for accurate cavitation predictions. The caveat is that they are less efficient at removing the initial transient solution, a part of the solution process that is sometimes the most time consuming. Another candidate approach is a geometric multigrid solution method.

Improved Models for Turbulent Flows:

It was observed that the applied turbulence model has a strong effect on the predicted cavity shape and stability. This implies the need for better modeling approaches. The real need is an embedded wall-function-based model of the interfacial boundary layers. Such a method would enable an improved representation of these boundary layers with a minimal number of cells.

Improved Methods for Design:

One of the major time consumers in ventilated cavitation analysis using a CFD method is determining the design conditions. A common specification is the cavitation number; the specific parameter that yields a cavity size that encloses a vehicle. Unfortunately, in a CFD method, the ventilation rate is specified and the cavitation
number is a result of the solution. Such a strategy creates an iterative analysis method. For simple geometries, $C_0-\sigma_c$ relations can be used, but they are only valid for configurations where a CFD-based analysis is only a slight improvement over semi-empirical methods. As already implemented in boundary element codes, future CFD strategies should focus on integrating a cavitation number specification into the CFD simulation.
Bibliography


27. Ubbink, O., Numerical prediction of two fluid systems with sharp interfaces, Department of Mechanical Engineering, Imperial College of Science, Technology, & Medicine, PhD Thesis, 1997.


32. Merkle, C., AERO/ME 527 Course notes, The Pennsylvania State University, University Park, PA.


83. Pressure and Lift Measurements on the Twist 11 Hydrofoil, Delft University of Technology, Internal report.

84. PEGASUS 5 Website, [http://people.nas.nasa.gov/~rogers/pegasus/intro.html](http://people.nas.nasa.gov/~rogers/pegasus/intro.html).


90. ImageMagick, www.imagemagick.org


Appendix A

Level-Set Implementation

Brief descriptions of the level set concepts and implementations are described to elucidate the implementation of the level set approach into the numerical formulation.

A.1 Derivation of Reinitialization

In the level-set method, rather than direct computation of the signed distance function a more efficient partial differential equation is solved. As mentioned in by Sethian [23], this is an observational-based relation. This section clarifies the basis of the reinitialization based on taking a differential form of Pythagoras’ Theorem. In the traditional reinitialization method, the signed distance is given by the solution of

\[
\frac{\partial \phi}{\partial \tau} = \text{sign}(\phi)(1 - |\nabla \phi|),
\]

A.1

This is based on the observation [23] that $|\nabla \phi| = 1$, or a special form of the eikonal equation, and the sign$(\phi)$ imposes boundary conditions that forces $\phi=0$ on the interface. The behavior of $|\nabla \phi| = 1$ is a differential form of Pythagoras’ Theorem, which is formally derived below. A diagram displaying the computation of the distance from the interface of point B, from information at point A, is presented in Fig. A-1. In this example, consider point B as being an infinitesimal offset from point A in both x and y.
For an infinitesimal change in $x$, $\Delta x$, with a corresponding change in the distance function, $\Delta \phi$, the Pythagorean Theorem suggests that

$$ (\phi + \Delta \phi)^2 = (x + \Delta x)^2 + (y)^2. \tag{A.2} $$

After expansion and linearization of $\Delta \phi$, i.e. $\Delta \phi = \Delta x \frac{\partial \phi}{\partial x}$, Eq. A.2 can be rewritten as

$$ \phi^2 + 2\phi \Delta x \frac{\partial \phi}{\partial x} + \Delta x^2 \left( \frac{\partial \phi}{\partial x} \right)^2 = x^2 + 2x \Delta x + \Delta x^2 + y^2 \tag{A.3} $$

Following some rearranging, the application of the Pythagorean Theorem $(\phi^2 - x^2 - y^2 = 0)$, and removing higher order terms, i.e. $\Delta x^2 \left( \frac{\partial \phi}{\partial x} \right)^2 \sim \Delta x^2 \ll \Delta x$ for small values of $\Delta x$, Eq. A.3 becomes

$$ \frac{x}{\phi} = \frac{\partial \phi}{\partial x}, \tag{A.4} $$

and in $y$ a similar relation is given as

$$ \frac{y}{\phi} = \frac{\partial \phi}{\partial y}. $$

Figure A-1: Diagram of the change in the distance function with respect to $x$ and $y$. 

![Diagram of the change in the distance function with respect to $x$ and $y$.](image-url)
Reapplying the Pythagorean Theorem once again, i.e. \( y^2 + x^2 = \phi^2 \), using the relations in Eq. A.4, a relation satisfying the distance function with changes in both the \( x \) and \( y \) directions is achieved as

\[
x^2 + y^2 = \phi^2 \left( \frac{\partial \phi}{\partial x} \right)^2 + \left( \frac{\partial \phi}{\partial y} \right)^2 \tag{A.5}
\]

Using \( y^2 + x^2 = \phi^2 \), then dividing \( \phi^2 \) through, and taking the square root of both sides, Eq. A.5 results in the differential form of the Pythagorean Theorem, or the basis of the signed-distance function reinitialization procedure, as

\[
\sqrt{\frac{x^2 + y^2}{\phi^2}} = \sqrt{\left( \frac{\partial \phi}{\partial x} \right)^2 + \left( \frac{\partial \phi}{\partial y} \right)^2} \implies 1 = |\nabla \phi| \tag{A.6}
\]

It is reiterated that the \( \text{sign}(\phi) \) term in Eq. A.1 enforces a boundary condition that forces the interface to retain a value of 0. The physical behavior is obvious in the one-dimensional case, where if the interface is assumed to occur at \( x=0 \), the behavior of the signed distance function, \( \phi \), as a function of \( x \), is \( |\phi(x)| = |x| \). Which implies that the gradient, or slope, magnitude is always unity.

**A.2 Expansions Used for Transport Equation Equivalence**

The expansions used for the transport equation transformations are given in this section. First, consider the transformation function given by

\[
H(\phi) = \frac{1}{2} \left( 1 + \tanh \left( \frac{\phi}{k\varepsilon} \right) \right) \text{ and } \phi(H) = k\varepsilon \tanh^{-1} (2H - 1). \tag{A.7}
\]
Furthermore, the Maclaurin expansion of the approximated Heaviside function is given as

\[ \tanh^{-1}(2H - 1) = (2H - 1) + \frac{(2H - 1)^3}{3} + \frac{(2H - 1)^5}{5} \ldots = \sum_{n=1}^{\infty} \frac{(2H - 1)^{2n-1}}{(2n-1)} , \quad A.8 \]

and can be substituted into the transport equation as

\[ \frac{D\phi}{Dt} = k \epsilon \frac{D}{Dt} \left[ \sum_{n=1}^{\infty} \frac{(2H - 1)^{2n-1}}{(2n-1)} \right] = 0 . \quad A.9 \]

To simplify the process, only consider the temporal derivative, however, no simplification is made that prevents the extension to spatial derivatives. Consider the form of the derivative given by

\[ \frac{\partial \phi}{\partial t} = k \epsilon \frac{\partial}{\partial t} \left[ \sum_{n=1}^{\infty} \frac{(2H - 1)^{2n-1}}{(2n-1)} \right] . \quad A.10 \]

First, it is useful to establish simplifications for the derivative terms. When the exponent (2n-1) is 1, the expansion simplifies as

\[ \frac{\partial}{\partial t} \left[ \frac{(2H - 1)^1}{(2n-1)} \right] = \frac{\partial}{\partial t} \left[ \frac{(2H)^1}{(2n-1)} \right] - \frac{\partial}{\partial t} \left[ \frac{(1)^1}{(2n-1)} \right] = 2 \frac{\partial H}{(2n-1) \partial t} . \quad A.11 \]

When the exponent is 2 and using the chain rule to expand the derivative term, the term simplifies as

\[ \frac{\partial}{\partial t} \left[ \frac{(2H - 1)^2}{(2n-1)} \right] = \frac{\partial}{\partial t} \left[ \frac{(2H - 1)(2H - 1)}{(2n-1)} \right] \]

\[ = (2H - 1) \frac{\partial}{\partial t} \left[ \frac{(2H - 1)}{(2n-1)} \right] + (2H - 1) \frac{\partial}{\partial t} \left[ \frac{(2H - 1)}{(2n-1)} \right] . \quad A.12 \]
Next, using the rule developed in Eq. 5.12, Eq. 5.22 simplifies to

$$\frac{\partial}{\partial t} \left[ \frac{(2H-1)^2}{(2n-1)} \right] = \frac{4(2H-1)}{(2n-1)} \frac{\partial H}{\partial t} \quad \text{(A.13)}$$

The next higher-order term, with an exponent of 3, reduces as

$$\frac{\partial}{\partial t} \left[ \frac{(2H-1)^3}{(2n-1)} \right] = \frac{\partial}{\partial t} \left[ \frac{(2H-1)(2H-1)^2}{(2n-1)} \right]$$

$$= \frac{\partial}{\partial t} \left[ \frac{(2H-1)^2}{(2n-1)} \right] + (2H-1)^2 \frac{\partial}{\partial t} \left[ \frac{(2H-1)}{(2n-1)} \right] \quad \text{(A.14)}$$

Again, reusing the rules developed in Eqns. 5.12 and A.13, Eq. 5.14 reduces to

$$\frac{\partial}{\partial t} \left[ \frac{(2H-1)^3}{(2n-1)} \right] = \frac{6(2H-1)^2}{(2n-1)} \frac{\partial H}{\partial t} \quad \text{(A.15)}$$

Lastly, the 4th order term reduces as

$$\frac{\partial}{\partial t} \left[ \frac{(2H-1)^4}{(2n-1)} \right] = \frac{\partial}{\partial t} \left[ \frac{(2H-1)(2H-1)^3}{(2n-1)} \right]$$

$$= \frac{\partial}{\partial t} \left[ \frac{(2H-1)^3}{(2n-1)} \right] + (2H-1)^3 \frac{\partial}{\partial t} \left[ \frac{(2H-1)}{(2n-1)} \right] \quad \text{(A.16)}$$

This time, reusing the rules developed in Eqns. 5.12 and 5.13, Eq. 5.17 reduces to

$$\frac{\partial}{\partial t} \left[ \frac{(2H-1)^4}{(2n-1)} \right] = \frac{8(2H-1)^3}{(2n-1)} \frac{\partial H}{\partial t} \quad \text{(A.17)}$$

Finally, the derivatives of the higher-order functions of $H$ can be expressed in terms of a scaled derivative of $H$. This is represented by the series given as
Consider how this can be applied to the transformation of the level-set function transport equation. From the rule developed in Eq. 5.16, and setting $m=2n-1$, the derivatives simplify to

$$\frac{\partial}{\partial t} \left[ \frac{(2H - 1)^1}{(2n-1)} \right] = \frac{2}{(2n-1)} \frac{\partial H}{\partial t}$$

$$\frac{\partial}{\partial t} \left[ \frac{(2H - 1)^2}{(2n-1)} \right] = \frac{4(2H - 1)}{(2n-1)} \frac{\partial H}{\partial t}$$

$$\frac{\partial}{\partial t} \left[ \frac{(2H - 1)^3}{(2n-1)} \right] = \frac{6(2H - 1)^2}{(2n-1)} \frac{\partial H}{\partial t}$$

$$\frac{\partial}{\partial t} \left[ \frac{(2H - 1)^4}{(2n-1)} \right] = \frac{8(2H - 1)^3}{(2n-1)} \frac{\partial H}{\partial t}$$

$$\frac{\partial}{\partial t} \left[ \frac{(2H - 1)^m}{(2n-1)} \right] = \frac{2m(2H - 1)^{m-1}}{(2n-1)} \frac{\partial H}{\partial t}.$$  \(A.18\)

$$\frac{\partial}{\partial t} \left[ \frac{(2H - 1)^{2n-1}}{(2n-1)} \right] = 2(2H - 1)^{2n-2} \frac{\partial H}{\partial t}. \quad \text{A.19}$$

These relations are used in Chapter 5 for transforming the variables of a transport equation.

### A.3 Implementation of the Level Set method Using a Volume-Fraction Formulation

The level-set method is implemented into the UNCLE-M numerical algorithm as described below. The only modification is in step 2.1.4 of the pseudo-time loop. This is performed using a single subroutine.
1 Initialize flow field: Note that here this is done with volume fractions rather than computing a distance to the interface through the domain.

2 Time step loop

2.1 Pseudo-Time Loop

2.1.1 Compute right and left hand side of linear system

2.1.2 Solve linear system

2.1.3 Integrate volume-fraction values in pseudo time → $\alpha^{n+1/2}$

2.1.4 Level-set Sharpening Procedure.

This step is performed periodically for steady integration and at the last pseudo-time step for unsteady integration.

**Method 1:** Signed-Distance Function

1) Transform the $n+1/2$ time level volume fraction to an approximated signed-distance function: $\phi^* = k\varepsilon \tanh^{-1} \left( 2\alpha^{n+1/2} - 1 \right)$

2) Apply a 4-Stage Runge-Kutta integration technique in $\tau$, the gradient term is computed using a van-Leer flux limited technique. Iterate, only a few iterations required. The sign($\phi$) approximation, $\left( 2\alpha^{n+1/2} - 1 \right)$ uses the $n+1/2$ time level so that the interface location does not move during the reinitialization procedure.

The solved equation is given as: $\frac{\partial \phi^*}{\partial \tau} = \left( 2\alpha^{n+1/2} - 1 \right) \left( 1 - |\nabla \phi^*| \right)$.

3) Recover the volume fraction: $\alpha^* = \frac{1}{2} \left( 1 + \tanh \left( \frac{\phi^*}{k\varepsilon} \right) \right)$
4) For stability purposes, updated volume fraction is only partially applied as:

\[ \alpha^{n+1} = \alpha^{n+1/2} + f_d \left( \alpha^* - \alpha^{n+1/2} \right) \]

The factor \( f_d \) is a number in the range of [0,1], where at \( f_d=0 \), the algorithm is shut off, and at \( f_d=1 \), is fully activated. This is likely an algorithm-dependent factor.

**Method 2:** Transformed Signed-Distance Function

1) Store the \( n+1/2 \) time level: \( \alpha^* = \alpha^{n+1/2} \)

2) Apply a 4-Stage Runge-Kutta integration technique in \( \tau \), the gradient term is computed using a van-Leer flux limited technique. Iterate, only a few iterations required. The sign(\( \phi \)) approximation, \( (2\alpha^{n+1/2} - 1) \) uses the \( n+1/2 \) time level so that the interface location does not move during the reinitialization procedure. The solved equation is given as:

\[ \frac{\partial \alpha^*}{\partial \tau} = (2\alpha^{n+1/2} - 1) \left( \Gamma^{-1} - |\nabla \alpha^*| \right) \]

Note that limiting of the residual, as:

\[ \Delta \tau \frac{\partial \alpha^*}{\partial \tau} = \min \left( \max \left( \Delta \tau \frac{\partial \alpha^*}{\partial \tau}, -\alpha^* \right), 1 - \alpha^* \right) \]

is useful to maintain a realizable solution.

3) For stability purposes, updated volume fraction is only partially applied as:

\[ \alpha^{n+1} = \alpha^{n+1/2} + f_d \left( \alpha^* - \alpha^{n+1/2} \right) \]

The factor \( f_d \) is a number in the range of [0,1], where at \( f_d=0 \), the algorithm is shut off, and at \( f_d=1 \), is fully activated. This is likely an algorithm-dependent factor.

**Method 3:** Realizable-Scaled Sharpening

1) Store the \( n+1/2 \) time level: \( \alpha^* = \alpha^{n+1/2} \)
2) Perform scaled and retain the realizable values of the volume fraction:

\[ \alpha^* = \min \left( \max \left( \frac{1}{2} \left[ \frac{\alpha^{n+1/2} - 0.5}{0.5 - \varepsilon_2} + 1 \right], 0 \right), 1 \right). \]

3) For stability purposes, updated volume fraction is only partially applied as:

\[ \alpha^{n+1} = \alpha^{n+1/2} + f_d \left( \alpha^* - \alpha^{n+1/2} \right) \]

The factor \( f_d \) is a number in the range of \([0,1]\), where at \( f_d = 0 \), the algorithm is shut off, and at \( f_d = 1 \), is fully activated. This is likely an algorithm-dependent factor.

2.1.5 Repeat (2.1) until converged in pseudo time

2.2 Repeat 2 for all time

3 Terminate program
Appendix B

Side Studies of Simulations

B.1 Mass Conservation for Confined, Unsteady Flows

For the simulation of unsteady problems, the pseudo-time terms must be small in order to conserve the volume of each species. Volume is considered here as this case is treated as incompressible, thus, volume conservations implies species mass conservation. These studies continue with a steady preconditioning approach. Fig. B-1 displays the total volume of air in a confined domain, for the evolution of a two-dimensional bubble flow driven by buoyancy. Several time-step sizes and values of $\beta$ are considered, where $\beta$ relates to the pseudo-sound speed ($U_\infty \beta^{-0.5}$ ~ Mach number is pseudo time). For a smaller $\beta$ values, little effect is observed when the time step size is changed. At a much larger $\beta$ value, however, volume conservation improves for a larger time step size. Thus, effectively removing the preconditioning seems to show some dependence on the issue.
Fig. B-2 examines the effects of integration effort in pseudo time on the conservation of mass of each species. These are all performed using a standard value of $\beta$ ($\beta=10U_{\infty}^2$). In Fig. B-2 (a), the effect of increasing the number of iterations in pseudo-time is displayed. The volume of air seems to be well conserved after about 100 iterations. This indicates that convergence in pseudo time is achieved quite slowly, for a steady preconditioner in a confined unsteady problem. A similar behavior was observed by Potsdam et al. [53]. With an increased number of iterations the convergence in pseudo time improves; one would expect that increasing the local time step size throughout the domain, would reduce the need for additional inner cycles. Such an effect is examined in Fig. B-2 (b), which shows no improvement in volume conservation. This shows strong support that mass conservation is only achieved through nonlinear integration.
B.2 Artificial Cavitation: Transition Effects

To examine the effect of modeling turbulence, a ventilated disk cavitator at $Re_N=1\times10^6$, $Fr_N=10.0$, is used. A summary of results are displayed in Fig. B-3. The comparison metrics are the cavitation number, maximum cavity radius, and the general predicted cavity shape. These comparisons are made using steady simulations, which are invalid when the cavity breaks down. The turbulence modeling strategy compares a fully turbulent case, a case where the boundary layer transition at the separation point on the cavitator, and a case without a turbulence model. Such a study will elicit the sensitivity to the turbulence model and to other modeling assumptions. Modeling a laminar cavitator
appears to have little effect on the initial state of the predicted cavity, possibly because the stagnation flow on the cavitator yields a very thin turbulent boundary layer. Experimental results [13] also show that the turbulent separation from the cavitator typically initiates a wavy cavity interface, which is a feature that is costly to resolve; this reduces the initial distinction between the laminar and turbulent solutions.

Figure B-3: Comparison of integral parameters for a supercavitating disk cavitator at $Fr_N=10.0$ and $Re_N=1\times10^6$, with fully turbulent, transitional, and laminar modeling assumptions.

The cavities display sensitivities to the turbulent versus non-turbulent simulations. Note that in a supercavititating flow, the cavities are typically smooth and glassy and on the water side of the cavity wall, and not typically turbulent. Whereas the air regions in the cavity are highly unsteady and likely turbulent, and with roughly one-thousandth the density, any air turbulence has little impact on the overall cavity shape predicted. It is expected that the turbulence model behavior will fail in the regions near the air-water interface. This is because there is no internal model to damp and control the behavior of this effectively resolved wall. It is apparent from the disk-cavitator simulations that the
turbulent results predict smaller cavities, with a different behavior in the breakdown of a stable cavity. Such sensitivities are suspect to general concerns in the turbulence modeling approach. The predicted cavity shapes are displayed in Fig. B-4. The most distinguished feature is the laminar case, where cavity dynamics are simulated rather than modeled (a steady form of MILES). This feature also results in the cavity breaking down earlier than the turbulent cavities, which possibly indicates the need to simulate rather than model, the interfacial instabilities.

**Figure B-4:** Comparisons of the predicted cavity shape with various turbulence model assumptions. Conditions correspond to $Fr_N=10.0$ and $Re_N=10^6$, the cavities assumed to be the $\alpha=0.5$ isosurfaces, which is colored by $1/|\nabla \alpha|$. The $C_Q$ values used are 1.5, 1.0, 0.5, 0.3, and 0.2 going from the upper to the lower cavities.

**B.3 Artificial Cavitation: Symmetry Plane Effects**

In many cases, it is beneficial to assume a plane of symmetry at the center of the body. That is for a supercavitating vehicle. This halves the cell requirement and could be
a beneficial assumption in terms of model reduction time. Recall that the cavity contains a highly sporadic flow field that is expected to have three-dimensional structures. The cavitation number and cavity size for the various ventilation rates, for the laminar and transitional cases, with and without assuming a symmetry plane is plotted in Fig. B-5. The cavitation number seems to be in good agreement with each other; however, the cavity size is smaller for the transitional case when using a symmetry plane at the lower cavitation numbers. The full 3D case also appears to maintain a cavity at slightly lower ventilation rates. Thus, the importance of modeling a full 3D geometry is important only in those ranges.

The assumption of a symmetry is further verified by evaluating the predicted cavity outlines. A comparison is displayed in Fig. B-6. Comparisons of the vertical and transverse cavity profiles are both displayed at a higher ventilation rate, where only small differences were observed in the cavitation number. There is little difference between the solutions indicating little sensitivity to the assumption.
B.4 Artificial Cavitation: Artificial Density Ratio Reductions

In general, the stability of the numerical algorithm strongly correlates to the density difference occurring between the modeled fluids across an interface. Here, an investigation of the sensitivity of the solution of the density of the ventilated gas for artificial supercavitation problems is investigated. The hope is that this is minimal, thus, the problems can be evaluated at lower density ratios and enable reduced simulation times. In an effort to retain the gravity-driven buoyancy effects, the true air density (roughly 1/1000 that of water) of the ventilated gas is used in the gravity term of Eq. 2.27. The density is then artificially increased in the remaining terms to improve the numerical stability. The idea is based on similar investigations of vaporous cavitation predictions about an O-give in Kunz et al. [24], where it was found that cavity size and

Figure B-6: Comparison of a predicted cavity with and without a symmetry plane. The conditions for this case are: $C_Q=1.5$, $Fr_N=10$, $Re_N=1\times10^6$. 
integral effects such as drag become independent of the density ratio after about a density ratio of 10.

The effect of the density of the ventilated gas into the cavity is displayed in Fig. B-7. In Fig. B-7 (a), is the effect of density on the predicted cavity pressure. The $C_Q$-$\sigma$ curve shows an impact with a varied air density primarily for the low-$\sigma$ cavities. Note that an inflection point occurs near $C_Q=1.0$, which is amplified for the higher density cases. The predicted cavity radius for the various air densities is displayed in Fig. B-7 (b). These results display that the radius is a strong function of the ventilated gas density. This trend appears to be continuing for even lower density ratios than examined. The cavity radius versus the cavitation number, for the varied density cases, is displayed in Fig. B-7 (c). For a given cavitation number, the cavity size appears to be in reasonable agreement. This seems to hold reasonably through an air density of 50.
Fig. B-8 displays the predicted cavity shapes for the density study. In general, the $C_Q$ cases of 1.5, 0.5, and 0.3 show little distinction at any density. This is, with the exception of the different cavity size which is difficult to discern visually. By inspection of the $C_Q=1.0$ cases in Fig. B-8, the cavities ventilated with $\rho=50$ and 100 gases appear to be in a pulsating mode (recall from Section 1.2.3) and are quite unsteady. For these cases, $\sigma Fr$ quantity is near unity, however, the value of $\beta$ is not different from the lower cases.
that do not exhibit this closure mode. Furthermore, the $C_q=0.2$ case with a density of 1 displays a very different closure than the higher density cases.

![Figure B-8: Disk cavitation at $Fr_D=10, Re_D=1\times10^6$, at various ventilation rates and for different densities of the noncondensable gas injected. The ventilation rates correspond to $C_Q$ values of 1.5, 1.0, 0.5, 0.3, and 0.2 from top to bottom for each of the densities given.](image)

These results indicate that although some integral quantities, such as lift and drag, can be achieved with quite different density ratios, other cavity features can change quite dramatically. These effects are important to aware of, especially when computing flows...
with body-filled cavities and when considering lower ventilation rates. Using such modifications can improve the stability and significantly reduce the run time. There does appear to be limits when using this approach in the transitional cavity closure regimes. These results also suggest that the true density of the air must be quite accurate in order to predict the correct cavity size.

### B.5 Evaluation of Oscillating Wing

The predictions of the lift are not in good agreement with the measured values from the experiment. This poor lift prediction is assumed to be a result of the poor baseline assumptions of the neglected transitional boundary layers, which, for this wing, the lift is highly sensitive to. This transitional effect is apparent in the baseline hydrofoil load predictions when accessed with the predictions using XFOIL [94]. XFOIL couples a panel method to a transition-modeling, integral boundary-layer method, and yields a quite accurate evaluation of the static hydrofoil loads. Predictions of the hydrofoil loads are displayed in Fig. B-9 for transitional and turbulent boundary layers. Note the strong influence of the boundary-layer properties that are apparent between the case where the transition is forced near the leading edge (similar to a fully turbulent simulation). When transition is allowed to naturally occur, the kink in the lift curve, at $c_l$ values just above 0.4, is where the laminar separation bubble appears to dominate the differences in the predicted loads approaching stall. Although the effect is not simulated in UNCLE-M as a result of the artificial alteration of the boundary layer through turbulence modeling.
assumptions, the predicted effects from XFOIL are used to correct the effect of the lack of modeled physics.

The two-dimensional predictions are corrected to correspond to the three-dimensional cases using Prandtl’s lifting-line theory. The required relations are given as

\[
C_L = \frac{c_{l,a} \alpha}{1 + \frac{c_{l,a}}{\pi AR(1 + \delta)}} = \frac{c_l(\alpha)}{1 + \frac{c_{l,a}}{\pi AR(1 + \delta)}},
\]

and

\[
C_D = c_d(\alpha) + \frac{C_L^2(1 + \delta)}{\pi AR},
\]

where for a nontapered wing with an \( AR = 4.0 \), the value of \( \delta \) is roughly 0.04. [95] Note that the \( c_{l,a} \alpha = c_l(\alpha) \) in this case because of the symmetric section. Any camber would prohibit this modification. Also, buoyancy is neglected as the span is positioned in the vertical direction for the experiments where this relation is applied.

Figure B-9: The predicted effect of modeling laminar flow for the investigated cross section.
The predicted forces in UNCLE-M are compared to the experiments and various Xfoil predictions with lifting-line theory in Fig. B-10 for both the XFOIL and UNCLE-M predictions. The fixed transition XFOIL predictions are closest to the measured experimental lift. It is unexpected that the transitional experiment is better predicted with a turbulent prediction; it is possible that the panel code has difficulties associated with the blunt trailing edge. The overall UNCLE-M lift predictions are not in good agreement with the experiment, which is expected as a result of the transition modeling deficit. Note the excellent agreement in predicted lift between the two-dimensional predictions, extended to three dimensional using lifting-line theory, and the three-dimensional CFD predictions. As expected, lifting-line theory extensions of a two-dimensional model are completely valid. With no experimental measurements of the drag, it is difficult to understand the accuracy of any of these the drag predictions. Some verification is observed in the fact that the three-dimensional UNCLE-M drag compares well to the Xfoil fixed-transition drag.
The sensitivity of this hydrofoil to transition is expected to be the cause of the poor lift predictions in the UNCLE-M simulations. The UNCLE-M lift predictions are corrected to account for transition based on differences from the fixed and naturally transitioning boundary layer predictions using Xfoil, which are plotted in Fig. B-11. Note that with these corrections, the lift is predicted to be in excellent agreement with the experiments. The result of this correction is somewhat a verification that the errors in the simulations are associated with poor boundary layer assumptions.

Figure B-10: Various prediction methods of the finite wing hydrodynamics.
B.6 Transitional Analysis of NACA 0009

Furthermore, as with most CFD codes, no adequate transition model is implemented. In the twisted wing results, transition to turbulence assumed to occur at 0.01c. This estimate is reasonable for the suction surface, where adverse pressure gradient create this transition, however, on the pressure surface, in the range of pitch angles examined, the transition occurs from x/c of 0.5 to 1.0. Transitional predictions are made using Xfoil, a potential-flow solver coupled to an integral boundary-layer method with a correlated e^n-based transition model capability. Here, the natural transition prediction is compared to another with transition fixed near the hydrofoil leading edge, which resembles that simulated in UNCLEM (transition at 0.01c). The effects are displayed in Fig. B-12. At low α values, the difference in drag is quite high and only when the drag component is dominated by pressure, the fully turbulent approximation is reasonable for drag predictions. Lift and pitching moment appears to be insensitive to transition. The other assumption, a direct result of fixing transition, is that all cases are independent of Re. All cases are run at a Re_c of 1.146e^6, corresponding to 6.97 m/s. This
argument is made by the fact that transition-sensitive attributes, that dominate the $Re$ dependence, are already removed with assumptions made in the turbulent behavior.

Figure B-12: Estimated effect of the transition location using Xfoil for the sectional hydrodynamic loads for the foil used for the Delft twisted wing [73].
Michael P. Kinzel was born in the town of Williams, AZ on June 21st, 1979. At the age of ten, he moved to Flagstaff, AZ and attended Thomas Elementary School, East Flagstaff Junior High, and then Coconino High School, where he graduated in 1997. He stayed in Flagstaff through his undergraduate studies where he pursued a B.S. in Mechanical Engineering at Northern Arizona University, graduating in May of 2001. While at Northern Arizona University, Michael worked his way through school as a painter for Daniel Rapier until he started CFD research with Dr. Duque. After finishing College, he worked as an intern for Enron Wind Energy and GE Wind, he then continued studying his M.S. in Aerospace Engineering at Penn State. This is where Michael studied active Gurney flaps, or MiTEs, working for Dr. Mark Maughmer at the Rotorcraft Center of Excellence until 2004. Later, he continued as a Ph.D. student in Aerospace Engineering working for Dr. Rob Kunz and Dr. Jay Lindau at the Applied Research Laboratory, where he was later hired on as a research faculty member in 2006. Following graduation, Michael’s plans are to continue his efforts in the Computational Mechanics Division at ARL.