Copyright 2005

by

Yogen Utturkar
To my wife and parents
ACKNOWLEDGMENTS

I am grateful to several individuals for their support in my dissertation work. The greater part of this work was made possible by the instruction of my teachers, and the love and support of my family and friends. It is with my heartfelt gratitude that I acknowledge each of them.

Firstly, I would like to express sincere thanks and appreciation to my advisor, Dr. Wei Shyy, for his excellent guidance, support, trust, and patience throughout my doctoral studies. I am very grateful for his remarkable wisdom, thought-provoking ideas, and critical questions during the course of my research work. I thank him for encouraging, motivating, and always prodding me to perform beyond my own limits. Secondly, I would like to express sincere gratitude towards my co-advisor, Dr. Nagaraj Arakere, for his firm support and caring attitude during some difficult times in my graduate studies. I also would like to express my appreciation to the members of my dissertation committee Dr. Louis Cattafesta, Dr. James Klausner, and Dr. Don Slinn, for their valuable comments and expertise to better my work. I deeply thank Dr. Siddharth Thakur (ST) for providing me substantial assistance with the STREAM code and for his cordial suggestions on research work and career planning.

My thanks go to all the members of our lab, with whom I have had the privilege to work. Due to the presence of all these wonderful people, work is more enjoyable. In particular, it was a delightful experience to collaborate with Jiongyang Wu, Tushar Goel, and Baoning Zhang on various research topics.
I would like to express my deepest gratitude towards my family members. My parents have always provided me unconditional love. They have always given top priority to my education, which made it possible for me to pursue graduate studies in the United States. I would like to thank my grandparents for their selfless affection and loving attitude during my early years. My wife’s parents and her sister’s family have been extremely supportive throughout my graduate education. I greatly appreciate their trust in my abilities.

Last but never least, I am thankful beyond words to my wife, Neeti Pathare. Together, we have walked through this memorable, cherishable, and joyful journey of graduate education. Her honest and unfltering love has been my most precious possession all these times. I thank her for standing besides me every time and every where. To Neeti and my parents, I dedicate this thesis!
TABLE OF CONTENTS

ACKNOWLEDGMENTS ........................................................................................................ iv
LIST OF TABLES ................................................................................................................ ix
LIST OF FIGURES ............................................................................................................... x
LIST OF SYMBOLS ........................................................................................................... xiv
ABSTRACT ...................................................................................................................... xviii

CHAPTER

1 INTRODUCTION AND RESEARCH SCOPE ................................................................. 1

1.1 Types of Cavitation ................................................................................................. 2
1.2 Cavitation in Cryogenic Fluids – Thermal Effect ................................................... 4
1.3 Contributions of the Current Study ...................................................................... 8

2 LITERATURE REVIEW .............................................................................................. 9

2.1 General Review of Recent Studies ......................................................................... 9
2.2 Modeling Thermal Effects of Cavitation .............................................................. 19
   2.2.1 Scaling Laws ................................................................................................... 19
   2.2.2 Experimental Studies ................................................................................... 24
   2.2.3 Numerical Modeling of Thermal Effects ...................................................... 27

3 STEADY STATE COMPUTATIONS ........................................................................... 33

3.1 Governing Equations ............................................................................................ 33
   3.1.1 Cavitation Modeling ...................................................................................... 35
      3.1.1.1 Merkle et al. Model .............................................................................. 35
      3.1.1.2 Sharp Interfacial Dynamics Model (IDM) ............................................. 35
      3.1.1.3 Mushy Interfacial Dynamics Model (IDM) .......................................... 37
   3.1.2 Turbulence Modeling ..................................................................................... 40
   3.1.3 Speed of Sound (SoS) Modeling .................................................................... 41
   3.1.4 Thermal Modeling ........................................................................................ 42
      3.1.4.1 Fluid property update .......................................................................... 42
      3.1.4.2 Evaporative cooling effects ................................................................. 43
3.1.5 Boundary Conditions.................................................................44
3.2 Results and Discussion ..............................................................45
  3.2.1 Cavitation in Non-cryogenic Fluids ....................................45
  3.2.2 Cavitation in Cryogenic Fluids ..............................................50
    3.2.2.1 Sensitivity analyses .......................................................52
    3.2.2.2 Assessment of cryogenic cavitation models over a wide range of
    conditions ................................................................................58
4 TIME-DEPENDENT COMPUTATIONS FOR FLOWS INVOLVING PHASE
  CHANGE.....................................................................................69
  4.1 Gallium Fusion ........................................................................71
    4.1.1 Governing Equations .......................................................72
    4.1.2 Numerical Algorithm .......................................................73
    4.1.3 Results ............................................................................78
      4.1.3.1 Accuracy and grid dependence ....................................79
      4.1.3.2 Stability ....................................................................82
      4.1.3.3 Data analysis by reduced-order description ..................83
  4.2 Turbulent Cavitating Flow under Cryogenic Conditions ............88
    4.2.1 Governing Equations .......................................................88
      4.2.1.1 Speed of sound modeling ...........................................89
      4.2.1.2 Turbulence modeling .................................................89
      4.2.1.3 Interfacial velocity model .........................................89
      4.2.1.4 Boundary conditions .................................................91
    4.2.2 Numerical Algorithm .......................................................91
    4.2.3 Results ............................................................................94
5 SUMMARY AND FUTURE WORK ...............................................101
  5.1 Summary .................................................................................101
  5.2 Future Work ..........................................................................103
APPENDIX
A BACKGROUND OF GLOBAL SENSITIVITY ANALYSIS ..................105
B REVIEW AND IMPLEMENTATION OF POD ...............................107
  B.1 Review ..................................................................................107
  B.2 Mathematical Background ....................................................110
  B.3 Numerical Implementation .....................................................112
    B.3.1 Singular Value Decomposition (SVD) ..............................112
    B.3.2 Post-processing the SVD Output ......................................113
  B.4 Flowchart .............................................................................114
  B.5 Computing Efforts ..............................................................114
LIST OF REFERENCES ................................................................................................. 116
BIOGRAPHICAL SKETCH .......................................................................................... 126
<table>
<thead>
<tr>
<th>Table</th>
<th>page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Properties of some cryogens in comparison to water at N.B.P, 1.01 bars</td>
<td>6</td>
</tr>
<tr>
<td>2. Source terms in cavitation models</td>
<td>13</td>
</tr>
<tr>
<td>3. Variants of the k-ε model</td>
<td>16</td>
</tr>
<tr>
<td>4. Summary of studies on thermal effects in cavitation</td>
<td>32</td>
</tr>
<tr>
<td>5. Flow cases chosen for the hydrofoil geometry</td>
<td>51</td>
</tr>
<tr>
<td>6. Flow cases chosen for the ogive geometry</td>
<td>51</td>
</tr>
<tr>
<td>7. Location of the primary vortex for the $St = 0.042$, $Ra = 2.2 \times 10^5$ and $Pr = 0.0208$ case</td>
<td>80</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Different types of cavitation (a) Traveling cavitation (b) Cloud cavitation (c) Sheet cavitation (d) Supercavitation (e) Vortex cavitation</td>
<td>3</td>
</tr>
<tr>
<td>2.</td>
<td>Saturation curves for water, Nitrogen, and Hydrogen</td>
<td>5</td>
</tr>
<tr>
<td>3.</td>
<td>Phasic densities along liquid-vapor saturation line for water and liquid Nitrogen</td>
<td>7</td>
</tr>
<tr>
<td>4.</td>
<td>General classification of numerical methods in cavitation</td>
<td>11</td>
</tr>
<tr>
<td>5.</td>
<td>Variation of Speed of Sound with phase fraction</td>
<td>14</td>
</tr>
<tr>
<td>6.</td>
<td>Two cavitation cases for $B$-factor analysis</td>
<td>21</td>
</tr>
<tr>
<td>7.</td>
<td>Schematic of bubble model for extracting speed of sound</td>
<td>27</td>
</tr>
<tr>
<td>8.</td>
<td>Schematic of cavity models (a) Distinct interface with vaporous cavity (Sharp IDM) (b) Smudged interface with mushy cavity (Mushy IDM)</td>
<td>36</td>
</tr>
<tr>
<td>9.</td>
<td>Behavior of $\rho_{l}/\rho_{-}$ and $\rho_{l}/\rho_{+}$ vs. $\alpha_{i}$ for the two models; $\rho_{l}/\rho_{+} = 100$ and $\beta = 0.09$</td>
<td>40</td>
</tr>
<tr>
<td>10.</td>
<td>Pressure-density and pressure-enthalpy diagrams for liquid nitrogen in the liquid-vapor saturation regime (Lemmon et al. 2002). Lines denote isotherms in Kelvin.</td>
<td>43</td>
</tr>
<tr>
<td>11.</td>
<td>Illustration of the computational domains for hemispherical projectile and NACA66MOD hydrofoil (non-cryogenic cases)</td>
<td>46</td>
</tr>
<tr>
<td>12.</td>
<td>Pressure coefficients over the hemispherical body ($\sigma = 0.4$); $D$ is the diameter of the hemispherical projectile. (a) Impact of grid refinement for Mushy IDM (b) Comparison between pressure coefficients of different models on the coarse grid.</td>
<td>47</td>
</tr>
<tr>
<td>13.</td>
<td>Cavity shapes and flow structure for different cavitation models on hemispherical projectile ($\sigma = 0.4$). (a) Merkle et al. Model (b) Sharp IDM (c) Mushy IDM</td>
<td>48</td>
</tr>
<tr>
<td>14.</td>
<td>Pressure coefficient over the NACA66MOD hydrofoil at two different cavitation numbers; $D$ is hydrofoil chord length. (a) $\sigma = 0.91$ (b) $\sigma = 0.84$</td>
<td>49</td>
</tr>
</tbody>
</table>
15. Illustration of the computational domain - accounting the tunnel - for the hydrofoil (Hord 1973a) and 0.357-inch ogive geometry (Hord 1973b) (cryogenic cases)........50

16. Non-cavitating pressure distribution (a) case ‘290C’, D represents hydrofoil thickness and x represents distance from the circular bend (b) case ‘312D’, D represents ogive diameter and x represents distance from the leading edge........52

17. Sensitivity of Merkle et al. Model prediction (surface pressure and temperature) to input parameters namely $C_{\text{dest}}$ and $C_{\text{prod}}$ for the hydrofoil geometry..................53

18. Main contribution of each design variable to the sensitivity of Merkle et al. (1998) model prediction; case ‘290C’ (a) Surface pressure (b) Surface temperature ..........55

19. Pressure and temperature prediction for Merkle et al. Model for the case with best match with experimental pressure; $C_{\text{dest}}^* = 0.85; t_x^* = 0.85; \rho_v^* = 1.1; L' = 0.9 .............56

20. Pressure and temperature prediction for Merkle et al. Model for the case with best match with experimental temperature; $C_{\text{dest}}^* = 1.15; t_x^* = 0.85; \rho_v^* = 1.1; L' = 1.1 .........56

21. Sensitivity of Mushy IDM prediction for case ‘290C’ (surface pressure and temperature) to the exponential transitioning parameter $\beta$........................................58

22. Surface pressure and temperature for 2-D hydrofoil for all cases involving liquid Nitrogen. The results referenced as ‘Mushy IDM’ and ‘Merkle et al. Model’ are contributions of the present study. .................................................................59

23. Cavitation number ($\sigma = \frac{p_v - p_s(T)}{0.5\rho U_{\infty}^2}$) based on the local vapor pressure – Merkle et al. Model. Note the values of $\sigma = \frac{p_v - p_s(T)}{0.5\rho U_{\infty}^2}$ for the cases ‘290C’ and ‘296B’ are 1.7 and 1.61, respectively. ........................................61


25. Evaporation ($\dot{m}^-$) and condensation ($\dot{m}^+$) source term contours between the two cavitation models – case ‘290C’. Refer to equations (3.8) and (3.18) for the formulations. .............................................................................................................63

26. Surface pressure and temperature for 2-D hydrofoil for cases involving liquid Hydrogen. The results referenced as ‘Mushy IDM’ and ‘Merkle et al. Model’ are contributions of the present study. .................................................................65

27. Surface pressure and temperature for axisymmetric ogive for all the cases (Nitrogen and Hydrogen). The results referenced as ‘Mushy IDM’ and ‘Merkle et al. Model’ are contributions of the present study.................................................................67
28. Schematic of the 2D Gallium square geometry with the Boundary Conditions (Shyy et al. 1998) ..........................................................................................................................71

29. 2D interface location at various instants for $St = 0.042$, $Ra = 2.2 \times 10^5$ and $Pr = 0.0208$. White circles represent interface locations obtained by Shyy et al. (1995) on a $41 \times 41$ grid at time instants at $t = 56.7$s, 141.8s, & 227s respectively. .79

30. Grid sensitivity for the $St = 0.042$, $Ra = 2.2 \times 10^5$ and $Pr = 0.0208$, 2D case (a) Centerline vertical velocity profiles at $t = 227$s (b) Flow structure in the upper-left domain at $t = 57$s; $41 \times 41$ grid (c) Flow structure in the upper-left domain at $t = 57$s; $81 \times 81$ grid. ..........................................................................................................................80

31. 3D interface location at $t = 57$s & 227$s for $St = 0.042$, $Ra = 2.2 \times 10^5$ and $Pr = 0.0208$ case on a $41 \times 41 \times 41$ grid. Top and bottom: adiabatic; North and West: $T = 0$; South and East: $T = 1$ (heated walls) ..........................................................................................................................82

32. Interface location and flow pattern for the 3D case, $St = 0.042$, $Ra = 2.2 \times 10^5$ and $Pr = 0.0208$ case at various $z$ locations, at $t = 227$s ..........................................................................................................................82

33. POD modes showing velocity streamlines $(\psi_i(r); i = 1,2,3,4)$ for $St = 0.042$, $Ra = 2.2 \times 10^3$ and $Pr = 0.0208$ case. $q(r,t) = \vec{V}(r,t)$. ..........................................................................................................................84

34 Scalar coefficients $(\phi_i(t); i = 1,2,...,8)$ for $St = 0.042$, $Ra = 2.2 \times 10^3$ and $Pr = 0.0208$ case. $q(r,t) = \vec{V}(r,t)$. ..........................................................................................................................85

35. Time instants when coefficients of respective POD modes show the first peak; $St = 0.042$, $Ra = 2.2 \times 10^5$ & $Pr = 0.0208$ case ..........................................................................................................................86

36. Horizontal-centerline vertical velocities for $St = 0.042$, $Ra = 2.2 \times 10^3$ and $Pr = 0.0208$ case at $t = 3$s ..........................................................................................................................86

37. Comparison between CFD solution and re-constructed solution for $St = 0.042$, $Ra = 2.2 \times 10^3$ and $Pr = 0.0208$ case at $t = 20$s. Contours represent velocity magnitude and lines represent streamlines ..........................................................................................................................87

38. Comparison between CFD solution and re-constructed solution for $St = 0.042$, $Ra = 2.2 \times 10^3$ and $Pr = 0.0208$ case at $t = 3$s. Contours represent velocity magnitude and lines represent streamlines ..........................................................................................................................88

39. Pressure history at a point near inlet, $\Delta = 0.11 D$, where $D$ is the chord length of the hydrofoil (a) ‘283B’ (b) ‘290C’ ..........................................................................................................................95

40. Cavity shape and flow structure for case ‘283B’. Lines denote streamlines ..........95

41. Pressure history at a point near the inlet for the case ‘290C’. $St$ denotes the non-dimensional perturbation frequency ($fD/U_\infty = f\bar{U}_\infty$) ..........................................................................................................................96
42. Pressure history at a point near the inlet for the cases ‘296B’ and ‘290C’. The non-dimensional perturbation frequency \( \left( \frac{fD}{U_{\infty}} = \frac{f}{f_{\infty}} \right) \) is 1.0..............................97

43 POD modes for the velocity field; \( \psi_i(r); i = 1,2,3; \ q(r,t) = \vec{V}(r,t) \) ...........................98

44 POD modes for the velocity field; \( \psi_i(r); i = 1,2,3; \ q(r,t) = \vec{V}(r,t) \) ...........................98
LIST OF SYMBOLS

\( \sigma \): cavitation number

\( \delta \): boundary layer/cavity thickness

\( L \): length (of cavity or cavitating object)

\( q \): heat flux; generalized flow variable in POD representation

\( V \): total volume

\( t \): time

\( x, y, z \): coordinate axes

\( r \): position vector

\( \xi \): streamwise direction in the curvilinear co-ordinate system

\( i, j, k, n \): indices

\( u, v, w \): velocity components

\( p \): pressure

\( T \): temperature

\( \rho \): density

\( \alpha \): volume fraction

\( f \): mass fraction

\( h \): sensible enthalpy

\( s \): entropy
$c$: speed of sound

$\gamma$: ratio of the two specific heats for gases

$\tau$: stress tensor

$P$: production of turbulent energy

$C_{\varepsilon_1}, C_{\varepsilon_2}, \sigma_k, \sigma_\varepsilon, C_\mu, C$: constants

$Q$: total kinetic energy (inclusive of turbulent fluctuations)

$k$: turbulent kinetic energy

$\varepsilon$: turbulent dissipation

$F$: filter function for filter-based modeling

$\mu$: dynamic viscosity

$K$: thermal conductivity

$\nu$: volume flow rate

$L$: latent heat

$C_p$: specific heat

$C_p$: pressure coefficient

$a$: thermal diffusivity

$\dot{m}$: volume conversion rate

$B$: B-factor to gauge thermal effect

$\Sigma$: dimensional parameter to assess thermal effect

$\beta$: control parameter in the Mushy IDM; coefficient of thermal expansion

$g$: gravitational acceleration

$U$: velocity scale

$D$: length scale (such as hydrofoil chord length or ogive diameter)
$R$: bubble radius

CFL: Courant, Freidricks, and Levy Number

St: Stefan Number

Ra: Rayleigh Number

Pr: Prandtl Number

Re: Reynolds Number

$\Delta$: difference; filter size in filter-based turbulence modeling

$\nabla$: gradient or divergence operator

$\psi$: POD mode

$\phi$: time-dependent coefficient in the POD series

$\eta$: energy content in the respective POD mode

**Subscripts:**

$\infty$: reference value (typically inlet conditions at the tunnel)

0: initial conditions

$s$: solid

$l$: liquid

$v$: vapor

$m$: mixture

$f$: friction

$Q$: discharge

$c$: cavity

$L$: laminar

$t$: turbulent
\( I \): interfacial

\( n \): normal to local gradient of phase fraction

\( sat \): saturation conditions

\( dest \): destruction of the phase

\( prod \): production of the phase

+: condensation

-: evaporation

\( A, H, S, M, G, B \): terms in a discretized equation

\( nb \): neighboring nodes

\( P \): at the cell of interest

**Superscripts/Overhead symbols:**

+: condensation

-: evaporation

‘: fluctuating component

*: normalized value; updated value in the context of PISO algorithm

\( \rightarrow \): vector

-: average

\( \sim \): Favre-averaged

\( n \): time step level

\( k \): iteration level
Abstract of Dissertation Presented to the Graduate School of the University of Florida in Partial Fulfillment of the Requirements for the Degree of Doctor of Philosophy

COMPUTATIONAL MODELING OF THERMODYNAMIC EFFECTS IN CRYOGENIC CAVITATION

By

Yogen Utturkar

August 2005

Chair: Wei Shyy
Cochair: Nagaraj Arakere
Major Department: Mechanical and Aerospace Engineering

Thermal effects substantially impact the cavitation dynamics of cryogenic fluids. The present study strives towards developing an effective computational strategy to simulate cryogenic cavitation aimed at liquid rocket propulsion applications. We employ previously developed cavitation and compressibility models, and incorporate the thermal effects via solving the enthalpy equation and dynamically updating the fluid physical properties. The physical implications of an existing cavitation model are reexamined from the standpoint of cryogenic fluids, to incorporate a mushy formulation, to better reflect the observed “frosty” appearance within the cavity. Performance of the revised cavitation model is assessed against the existing cavitation models and experimental data, under non-cryogenic and cryogenic conditions.

Steady state computations are performed over a 2D hydrofoil and an axisymmetric ogive by employing real fluid properties of liquid nitrogen and hydrogen. The thermodynamic effect is demonstrated under consistent conditions via the reduction in
the cavity length as the reference temperature tends towards the critical point. Justifiable agreement between the computed surface pressure and temperature, and experimental data is obtained. Specifically, the predictions of both the models are better; for the pressure field than the temperature field, and for liquid nitrogen than liquid hydrogen. Global sensitivity analysis is performed to examine the sensitivity of the computations to changes in model parameters and uncertainties in material properties.

The pressure-based operator splitting method, PISO, is adapted towards typical challenges in multiphase computations such as multiple, coupled, and non-linear equations, and sudden changes in flow variables across phase boundaries. Performance of the multiphase variant of PISO is examined firstly for the problem of gallium fusion. A good balance between accuracy and stability is observed. Time-dependent computations for various cases of cryogenic cavitation are further performed with the algorithm. The results show reasonable agreement with the experimental data. Impact of the cryogenic environment and inflow perturbations on the flow structure and instabilities is explained via the simulated flow fields and the reduced order strategy of Proper Orthogonal Decomposition (POD).
CHAPTER 1
INTRODUCTION AND RESEARCH SCOPE

The phenomenon by which a liquid forms gas-filled or vapor-filled cavities under the effect of tensile stress produced by a pressure drop below its vapor pressure is termed cavitation (Batchelor 1967). Cavitation is rife in fluid machinery such as inducers, pumps, turbines, nozzles, marine propellers, hydrofoils, journal bearings, squeeze film dampers etc. due to wide ranging pressure variations along the flow. This phenomenon is largely undesirable due to its negative effects namely noise, vibration, material erosion etc. Detailed description of these effects can be readily obtained from literature. It is however noteworthy that the cavitation phenomenon is also associated with useful applications. Besides drag reduction efforts (Lecoffre 1999), biomedical applications in drug delivery (Ohl et al. 2003) and shock wave lithotripsy (Tanguay and Colonuis 2003), environmental applications for decomposing organic compounds (Kakegawa and Kawamura 2003) and water disinfection (Kalamuck et al. 2003), and manufacturing and material processing applications (Soyama and Macodiyo 2003) are headed towards receiving an impetus from cavitation. Comprehensive studies on variety of fluids such as water, cryogens, and lubricants have provided significant insights on the dual impact of cavitation. Experimental research methods including some mentioned above have relied on shock waves (Ohl et al. 2003), acoustic waves (Chavanne et al. 2002), and laser pulses (Sato et al. 1996), in addition to hydrodynamic pressure changes, for triggering cavitation. Clearly, research potential in terms of understanding the mechanisms and characteristics of cavitation in different fluids, and their applications and innovation is
tremendous. The applicability and contributions of the present study within the above-mentioned framework are described later in this chapter.

1.1 Types of Cavitation

Different types of cavitation are observed depending on the flow conditions and fluid properties. Each of them has distinct characteristics as compared to others. Five major types of cavitation have been described in literature. They are as follows:

(a) Traveling cavitation

It is characterized by individual transient cavities or bubbles that form in the liquid, expand or shrink, and move downstream (Knapp et al. 1970). Typically, it is observed on hydrofoils at small angles of attack. The density of nuclei present in the upstream flow highly affects the geometries of the bubbles (Lecoffre 1999). Traveling cavitation is illustrated in Figure 1(a).

(b) Cloud cavitation

It is produced by vortex shedding in the flow field and is associated with strong vibration, noise, and erosion (Kawanami et al. 1997). A re-entrant jet is usually the causative mechanism for this type of cavitation (Figure 1(b)).

(c) Sheet cavitation

It is also known as fixed, attached cavity, or pocket cavitation (Figure 1(c)). Sheet cavitation is stable in quasi-steady sense (Knapp et al. 1970). Though the liquid-vapor interface is dependent on the nature of flow, the closure region is usually characterized by sharp density gradients and bubble clusters (Gopalan and Katz 2000).

(d) Supercavitation

Supercavitation can be considered as an extremity of sheet cavitation wherein a substantial fraction of the body surface is engulfed by the cavity (Figure 1(d)). It is
observed in case of supersonic underwater projectiles, and has interesting implications on viscous drag reduction (Kirschner 2001).

(e) Vortex cavitation

It is observed in the core of vortices in regions of high shear (Figure 1(e)). It mainly occurs on the tips of rotating blades and in the separation zone of bluff bodies (Knapp et al. 1970).

Figure 1. Different types of cavitation\textsuperscript{*} (a) Traveling cavitation (b) Cloud cavitation (c) Sheet cavitation (d) Supercavitation (e) Vortex cavitation

\textsuperscript{*} Reproduced from Franc et al. (1995) with permission from EDP Sciences
1.2 Cavitation in Cryogenic Fluids – Thermal Effect

Cryogens serve as popular fuels for the commercial launch vehicles while petroleum, hypergolic propellants, and solids are other options. Typically, a combination of liquid oxygen (LOX) and liquid hydrogen (LH2) is used as rocket propellant mixture. The boiling points of LOX and LH2 under standard conditions are -183 F and -423 F, respectively. By cooling and compressing these gases from regular conditions, they are stored into smaller storage tanks. The combustion of LOX and LH2 is clean since it produces water vapor as a by-product. Furthermore, the power/gallon ratio of LH2 is high as compared to other alternatives. Though storage, safety, and extreme low temperature limits are foremost concerns for any cryogenic application, rewards of mastering the use of cryogens as rocket propellants are substantial (NASA Online Facts 1991).

A turbopump is employed to supply the low temperature propellants to the combustion chamber which is under extremely high pressure. An inducer is attached to the turbopump to increase its efficiency. Design of any space vehicle component is always guided by minimum size and weight criteria. Consequently, the size constraint on the turbopump solicits high impellor speeds. Such high speeds likely result in a zone of negative static pressure (pressure drop below vapor pressure) causing the propellant to cavitate around the inducer blades (Tokumasu et al. 2002). In view of the dire consequences, investigation of cavitation characteristics in cryogens, specifically LOX and LH2, is an imperative task.

Intuitively, physical and thermal properties of a fluid are expected to significantly affect the nature of cavitation. For example, Helium-4 shows anomalous cavitation properties especially past the $\lambda$-point temperature mainly due to its transition to superfluidity (Daney 1988). Besides, quantum tunneling also attributes to cavity
formation in Helium-4 (Lambare et al. 1998). Cavitation of Helium-4 in the presence of a glass plate (heterogenous cavitation) has lately produced some unexpected results (Chavanne et al. 2002), which are in contrast to its regular cavitating pattern observed under homogenous conditions (without a foreign body). Undoubtedly, a multitude of characteristics and research avenues are offered by different types of cryogenic fluids. The focus of the current study is, however, restricted to cryogenic fluids such as LOX, LH2, and liquid Nitrogen due to their aforesaid strong relevance in space applications. It is worthwhile at the outset to contrast their behavior/physical properties to water.

![Saturation curves for water, Nitrogen, and Hydrogen](image)

Figure 2. Saturation curves for water, Nitrogen, and Hydrogen

---

* Obtained from REFPROP v 7.0 by Lemmon et al. (2002)
Table 1. Properties of some cryogens in comparison to water at N.B.P, 1.01 bars

<table>
<thead>
<tr>
<th>Substance</th>
<th>Specific heat (J/Kg.K)</th>
<th>Liquid density (kg/m³)</th>
<th>Liquid/Vapor density</th>
<th>Thermal conductivity (W/mK)</th>
<th>Vaporization heat (KJ/Kg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water</td>
<td>4200</td>
<td>958</td>
<td>1603</td>
<td>681</td>
<td>2257</td>
</tr>
<tr>
<td>H₂</td>
<td>9816</td>
<td>71</td>
<td>53</td>
<td>100</td>
<td>446</td>
</tr>
<tr>
<td>N₂</td>
<td>2046</td>
<td>809</td>
<td>175</td>
<td>135</td>
<td>199</td>
</tr>
<tr>
<td>O₂</td>
<td>1699</td>
<td>1141</td>
<td>255</td>
<td>152</td>
<td>213</td>
</tr>
</tbody>
</table>

Source: Weisend et al. (1998)

Refer to Figure 2 and Table 1. The operating point of cryogenic liquids is generally quite close to the critical point unlike water. Furthermore, as indicated by Figure 2, the saturation pressure curves for cryogens demonstrate a much steeper slope v/s temperature, as compared to water. Consequently, the vapor pressure of liquids such as LH₂ and liquid Nitrogen is expected to show great sensitivity to small temperature drops. Since cavitation is predominantly governed by the vapor pressure, the significance of this thermodynamic sensitivity on the flow problem is clear upfront.

Further insight can be obtained from Table 1. Liquid-to-vapor density ratio in the case of cryogenic fluids is substantially lower than water. Thus, these fluids require a greater amount of liquid, and in turn latent heat, than water to cavitate (form vapor) under similar flow conditions. Furthermore, the thermal conductivity for the low temperature fluids is consistently lower than water. These facts indicate that the sensible-latent heat conversion in cryogenic fluids is expected to develop a noticeable temperature gradient surrounding the cavitation region. The impact of this local temperature drop is magnified by the steep saturation curves observed in Figure 2. Subsequently, the local vapor pressure experiences a substantial drop in comparison to the freestream vapor pressure leading to suppression in the cavitation intensity. Experimental (Hord 1973a, 1973b) and
numerical results (Deshpande et al. 1997) on sheet cavitation have shown a 20-40% reduction in cavity length due to the thermodynamic effects in cryogenic fluids.

Figure 3. Phasic densities along liquid-vapor saturation line for water and liquid Nitrogen

Additionally, the physical properties of cryogenic fluids, other than vapor pressure, are also thermo-sensible, as illustrated in Figure 3. Thus, from a standpoint of numerical computations, simulating cryogenic cavitation implies a tight coupling between the non-linear energy equation, momentum equations, and the cavitation model, via the iterative update of fluid properties (such as vapor pressure, densities, specific heat, thermal conductivity, viscosity etc.) with changes in the local temperature. Encountering these difficulties is expected to yield a numerical methodology specifically well-suited for cryogenic cavitation, and forms the key emphasis of the present study.

\footnote{Obtained from REFPROP v 7.0 by Lemmon et al. (2002)}
1.3 Contributions of the Current Study

The major purpose of the present study is to develop a robust and comprehensive computational tool to simulate cavitating flow under cryogenic conditions. The specific contributions of the endeavor are summarized as follows:

(a) A review of the experimental and computational studies on cryogenic cavitation

(b) Coupling of energy equation to the existing cavitation framework in conjunction with iterative update of the real fluid properties with respect to the local temperature

(c) Adaptation of an existing cavitation model (Senocak and Shyy 2004a, 2004b) to accommodate the physics of the mushy nature of cavitation observed in cryogenic fluids (Hord 1973, Sarosdy and Acosta 1961)

(d) Demonstration of the impact of thermodynamic effects on cavitation over wide-ranging temperatures, for two different cryogenic fluids. Assessment of the computational framework alongside available experimental and numerical data.

(e) Global sensitivity analysis of the computational predictions (pressure and temperature) with respect to the cavitation model parameters and the temperature-dependent material properties, via employing the response surface approach.

(f) Adaptation of the pressure-based operator splitting method, PISO (Issa 1985), to multiphase environments typically characterized by strong interactions between the governing equations and steep variations of flow variables across the phase boundary

(g) Assessment of the stability and accuracy of the non-iterative algorithm (PISO variant) on the test problem of Gallium fusion.

(h) Time-dependent computations of cryogenic cavitation (with the PISO variant) by applying perturbation to the inlet temperature

(i) Employment of Proper Orthogonal Decomposition (POD) to offer a concise representation to the simulated CFD data.
CHAPTER 2
LITERATURE REVIEW

Cavitation has been the focal point of numerous experimental and numerical studies in the area of fluid dynamics. A review of these studies is presented in this chapter. Since cryogenic cavitation remains the primary interest of this study, the general review on cavitation studies is purely restricted according to relevance. Specifically, the numerical approaches in terms of cavitation, compressibility, and turbulence modeling are briefly reviewed in the earlier section with reference to pertinent experiments. The later section mainly delves into the issues of thermal effects of cavitation. Current status of numerical strategies in modeling cryogenic cavitation, and their merits and limitations are reported to underscore the gap bridged by the current research study.

2.1 General Review of Recent Studies

Computational modeling of cavitation has complemented experimental research on this topic for a long time. Some earlier studies (Reboud et al. 1990, Deshpande et al. 1994) relied on potential flow assumption (Euler equations) to simulate flow around the cavitating body. However, simulation strategies by solving the Navier-Stokes equations have gained momentum only in the last decade. Studies in this regard can be broadly classified based on their interface capturing method. Chen and Heister (1996) and Deshpande et al. (1997) adopted the interface tracking Marker and Cell approach in their respective studies, which were characterized by time-wise grid regeneration and the constant cavity-pressure assumption. The liquid-vapor interface in these studies was explicitly updated at each time step by monitoring the surface pressure, followed by its
reattachment with an appropriate wake model. This effort was mainly well-suited for only sheet cavitation. The second category, which is the homogeneous flow model, has been a more popular approach, wherein the modeling for both phases is adopted via a single-fluid approach. The density change over the interface is simply modeled by a liquid mass fraction \((f_i)\) or a liquid volume fraction \((\alpha_i)\) that assumes values between 0 and 1. The mixture density can be expressed in terms of either fraction as follows:

\[
\rho_m = \alpha_i \rho_l + (1-\alpha_i) \rho_v \tag{2.1}
\]

\[
\rho_m = \frac{\rho_l}{f_i} + \frac{\rho_v}{1-f_i} \tag{2.2}
\]

The precise role of various cavitation models, which are reviewed later, is prediction of this volume/mass fraction as a function of space and time.

Both, density-based and pressure-based methods have been successfully adopted in conjunction with the single-fluid method in numerous studies. Due to unfeasibility of LES or DNS methods for multiphase flow, RANS approach through \(k-\varepsilon\) turbulence model has been mostly employed in the past studies. The main limitation of density-based methods (Merkle et al. 1998, Ahuja et al. 2001, Lindau et al. 2002, Iga et al. 2003) is requirement of pre-conditioning (Kunz et al. 2000) or the artificial density approach for flows which may be largely incompressible. Pressure-based methods (Ventikos and Tzabiras 2000, Athavale et al. 2001, Senocak and Shyy 2002, Singhal et al. 2002) on the other hand are applicable over a wide range of Mach numbers. Modeling the speed of sound in the mixture region and computational efficiency for unsteady calculations are the main issues for pressure-based solvers. A broad classification of the numerical methods is illustrated in Figure 4.
Figure 4. General classification of numerical methods in cavitation

Within the broad framework of above methods, several cavitation, compressibility, and turbulence models have been deployed, which are reviewed next as per the aforesaid order.

As mentioned earlier, the role of cavitation modeling is basically determination of the phase fraction. Different ideas have been proposed to generate the variable density field. Some studies solved the energy equation and obtained the density either by employing an equation of state (EOS) (Delannoy and Kueny 1990, Edwards et al. 2000) or from thermodynamic tables (Ventikos and Tzabiras 2000). Saurel et al. 1999 developed a cavitation model for hypervelocity underwater projectiles based on separate EOS for the liquid, vapor and mixture zones. EOS approach in first place does not capture the essential cavitation dynamics due to its equilibrium assumption in the phase change process. Furthermore, prevalence of isothermal condition in case of non-
thermosensible fluids such as water imparts a barotropic form, $\rho_m = f(p)$, to the EOS. Thus, cavitation modeling \textit{via} employing the EOS is devoid of the capability to capture barotropic vorticity generation in the wake region as demonstrated by Gopalan and Katz (2000). Transport-equation based cavitation models in contrast overcome the above limitations and are more popular in research studies. Typically, this approach determines the liquid volume fraction ($\alpha$) or the vapor mass fraction ($f_v$) by solving its transport equation as shown below.

\[
\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \vec{u}) = \dot{m}^+ + \dot{m}^- \tag{2.3}
\]

\[
\frac{\partial \rho_{mf_v}}{\partial t} + \nabla \cdot \left( \rho_{mf_v} \vec{u} \right) = \dot{m}^+ + \dot{m}^- \tag{2.4}
\]

Formulation of the source terms shown in above equation(s) constitutes the major effort in model development. Singhal et al. (1997), Merkle et al. (1998), Kunz et al. (2000), and Singhal et al. (2002) formulated these source terms strongly based on empirical judgment. However, Senocak and Shyy (2002, 2004a) developed a cavitation model fundamentally relying on interfacial mass and momentum transfer. Though their model was not completely empiricism-free, it transformed the empirical coefficients used in the earlier models into a physically explicable form. The ability of the model to capture the barotropic vorticity, $\nabla p \times \nabla \frac{1}{\rho}$, in the closure region was also clearly demonstrated by Senocak and Shyy (2002, 2004a). The source terms of each of the above models along with value of empirical constants are tabulated in Table 2.
### Table 2 Source terms in cavitation models

<table>
<thead>
<tr>
<th>Authors</th>
<th>$m^+$ - Production Term</th>
<th>$m^-$ - Destruction term</th>
</tr>
</thead>
<tbody>
<tr>
<td>Singhal et al. (1997)</td>
<td>$C_{prod} \frac{\text{MAX} (p - p_v, 0) (1 - \alpha_i)}{(0.5 \rho U_z^2) \rho_f} \times 8 \times 10^3$</td>
<td>$C_{dest} \rho_i \frac{\text{MIN} (p - p_v, 0) \alpha_i}{(0.5 \rho U_z^2) \rho_f} \times C_{dest} = 1$</td>
</tr>
<tr>
<td>Merkle et al. (1998)</td>
<td>$C_{prod} = 8 \times 10^3$</td>
<td></td>
</tr>
<tr>
<td>Kunz et al. (2000)</td>
<td>$C_{prod} = 3 \times 10^4$</td>
<td></td>
</tr>
<tr>
<td>Singhal et al. (2002)</td>
<td>$C_{prod} \frac{U_z}{\gamma} \frac{\rho_i \rho_f}{\rho_f} [\frac{2}{3} \text{MAX} (p - p_v, 0)]^{1/2}$</td>
<td>$C_{dest} \frac{U_z}{\gamma} \frac{\rho_i \rho_f}{\rho_f} [\frac{2}{3} \text{MIN} (p - p_v, 0)]^{1/2}$</td>
</tr>
<tr>
<td></td>
<td>$C_{prod} / \gamma = 3.675 \times 10^3$</td>
<td>$C_{dest} / \gamma = 1.225 \times 10^3$</td>
</tr>
<tr>
<td>Senocak and Shyy (2002, 2004a)</td>
<td>$C_{prod} \frac{\text{MAX} (p - p_v, 0) (1 - \alpha_i)}{(0.5 \rho U_z^2) \rho_f} \times \frac{1}{(\rho_i - \rho_v)(U_{v,n} - U_{l,n})^2}$</td>
<td>$C_{dest} \rho_i \frac{\text{MIN} (p - p_v, 0) \alpha_i}{(0.5 \rho U_z^2) \rho_f} \times \frac{1}{(\rho_i - \rho_v)(U_{v,n} - U_{l,n})^2}$</td>
</tr>
</tbody>
</table>

As clearly seen from the above table, the cavitation model of Senocak and Shyy (2002) is consistent with that of Singhal et al. (1997) and Merkle et al. (1998). However, the model constants have assumed physicality. The terms $U_{l,n}$ & $U_{v,n}$, which represent the liquid-vapor interface velocity and the normal component of interfacial vapor velocity are calculated by suitable approximations (Senocak and Shyy 2002, 2004a).

Numerous experimental studies (Leighton et al. 1990, Clarke and Leighton 2000) addressing compressible bubble oscillations are available. Muzio et al. (1998) developed a numerical compressibility model for acoustic cavitation in a bubble inclusive of viscosity and surface tension effects. However, these models are difficult to deploy for practical multiphase computations.
Figure 5 (adapted from Hosangadi et al. 2002) illustrates the modeled behavior of speed of sound v/s the phase fraction. As observed from the figure, the speed of sound in the biphasic mixture can be 2-3 orders of magnitude lower than the individual phases.

![Figure 5. Variation of Speed of Sound with phase fraction](image)

Consequently, a bulk incompressible flow may transform into a transonic or supersonic stage locally in the mixture region. Due to lack of dependable equation of state for multiphase mixtures, modeling sound propagation, which is an imperative issue in the numerical computations, is still an open question. A closed form expression for the speed of sound in the mixture region may be obtained by eigenvalue analysis on the strongly conservative form of the governing equations (Hosangadi et al. 2002), as shown below.

\[
\frac{1}{c_m} = \rho_m \left[ \frac{\alpha_v}{\rho_v a_v^2} + \frac{\alpha_i}{\rho_i a_i^2} \right]
\]  

(2.5)

Venkateswaran et al. (2002) proposed use of perturbation theory to obtain an efficient pre-conditioned system of equations, which were consistent in the incompressible as well as compressible regime. Improvement in cavitation dynamics by accounting the compressibility effects was reported. The incorporation of Speed of Sound (SoS) model
into pressure-based cavitation computations was significantly advanced by Senocak and Shyy (2002, 2003). In pressure-based solvers, the SoS model affects the solution mainly through the pressure correction equation. The following relationship is adopted between the density correction and the pressure correction terms, while enforcing the mass-conservation treatment through the pressure correction equation:

\[ \rho^{'\prime} m = C_\rho p^\prime \]  

(2.6)

The implementation of equation (2.6) imparts a convective-diffusive form to the pressure equation. Two SoS models were proposed by Senocak and Shyy (2003, 2004a, 2004b).

SoS-1: \[ C_\rho = \frac{\partial \rho}{\partial p} = C(1 - \alpha_i) \]  

(2.7)

SoS-2: \[ C_\rho = \left( \frac{\Delta \rho}{\Delta p} \right) = \left( \frac{\partial \rho}{\partial p} \right) = \frac{\rho_{i+1} - \rho_{i-1}}{p_{i+1} - p_{i-1}} \]  

(2.8)

While SoS-1 is a suitable approximation to the curve shown in Figure 5, SoS-2 approximates the fundamental definition of speed of sound by adopting a central-difference spatial derivative along the streamline direction (\( \xi \)) instead of differentiating along the isentropic curve. Computations by Senocak and Shyy (2003) on convergent-divergent nozzle demonstrated far better capability of SoS-2 to mimic the transient behavior observed in experiments. Wu et al. (2003b) extended the model assessment by pointing out the dramatic time scale differences between the two models. This points the fact that compressibility modeling is a sensitivity issue and must be handled carefully.

RANS-based approaches in form of two-equation \( k - \varepsilon \) models have been actively pursued to model turbulent cavitating flows. The \( k \) and \( \varepsilon \) transport equations along with the definition of turbulent viscosity are summarized below.
The production of turbulent kinetic energy \( P_t \) is defined as:

\[
\frac{\partial (\overline{p}_m k)}{\partial t} + \frac{\partial (\overline{p}_m \overline{u}_j k)}{\partial x_j} = P_t - \overline{p}_m \varepsilon + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_s}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \tag{2.9}
\]

while, the turbulent viscosity is defined as:

\[
\frac{\partial (\overline{p}_m \varepsilon)}{\partial t} + \frac{\partial (\overline{p}_m \overline{u}_j \varepsilon)}{\partial x_j} = C_{e1} \frac{\varepsilon}{k} P_t - C_{e2} \overline{p}_m \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_s}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \tag{2.10}
\]

The model coefficients, namely, \( C_{e1}, C_{e2}, \sigma_k \) and \( \sigma_\varepsilon \) have three known non-trivial variants which have been summarized in Table 3.

<table>
<thead>
<tr>
<th>Authors</th>
<th>( C_{e1} ) and Relevant Details</th>
<th>( C_{e2} )</th>
<th>( \sigma_k )</th>
<th>( \sigma_\varepsilon )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Launder and Spalding (1974)</td>
<td>1.44</td>
<td>1.92</td>
<td>1.3</td>
<td>1.0</td>
</tr>
<tr>
<td>Shyy et al. (1997)</td>
<td>( 1.15 + 0.25 \frac{P_t}{\varepsilon} )</td>
<td>1.9</td>
<td>1.15</td>
<td>0.89</td>
</tr>
<tr>
<td>Younis (2003) (personal communication)</td>
<td>( (1.15 + 0.25 \frac{P_t}{\varepsilon}) \times (1 + 0.38 \frac{k}{\varepsilon} \frac{\partial Q}{\partial t} / Q) )</td>
<td>1.9</td>
<td>1.15</td>
<td>0.89</td>
</tr>
<tr>
<td>Johansen et al. (2004)</td>
<td>1.44</td>
<td>1.92</td>
<td>1.3</td>
<td>1.0</td>
</tr>
</tbody>
</table>

While the Launder and Spalding (1974) model is calibrated for equilibrium shear flows, the model by Shyy et al. (1997) accommodates non-equilibrium effects by introducing a
subtle turbulent time scale into $C_{e_1}$. The RANS model of Younis (2003, personal communication), in comparison, accounts for the time history effects of the flow. Wu et al. (2003a, 2003b) assessed the above RANS models on turbulent cavitating flow in a valve. Experimental visuals of the valve flow (Wang 1999) have demonstrated cavitation instability in form of periodic cavity detachment and shedding. However, Wu et al. (2003b) reported that the $k - \varepsilon$ model over predicts the turbulent viscosity and damps such instabilities. Consequently, the RANS computations were unable to capture the shedding phenomenon, and also showed restrained sensitivity to the above variants of the $k - \varepsilon$ model. From standpoint of an alternate approach, the impact of filter-based turbulence modeling on cavitating flow around a hydrofoil was reported (Wu et al. 2003c). The filter-based model relies on the two-equation formulation and uses identical coefficient values as proposed by Launder and Spalding (1974), but imposes a filter on the turbulent viscosity as seen below.

\begin{equation}
\mu_t = \frac{\rho \mu C_{\mu} k^2}{\varepsilon} F, \quad C_{\mu} = 0.09
\end{equation}

The filter function ($F$) is defined in terms of filter size ($\Delta$) as:

\begin{equation}
F = \text{Min}[1, C_{\Delta} \frac{\Delta \varepsilon}{k^{3/2}}]; \quad C_{\Delta} = 1
\end{equation}

Note that the forms of viscosity in equations (2.12) and (2.13) are comparable barring the filter function. The proposed model recovers the Launder and Spalding (1974) model for coarse filter sizes. Furthermore, at near-wall regions, the imposed filter value $F = 1$ enables the use of wall functions to model the shear layer. However, in the far field zone if the filter size is able address the turbulent length scale $\frac{k^{3/2}}{\varepsilon}$, the solution is computed
directly (\( \mu_t \approx 0 \)). The filter-based model is also characterized by the independence of the filter size from the grid size. This model enhanced the prediction of flow structure for single-phase flow across a solid cylinder (Johansen et al. 2004). Wu et al. (2003c) reported substantial unsteady characteristics for cavitating flow around a hydrofoil because of this newly developed model. Furthermore, the time-averaged results of Wu et al. (2003c) (surface pressure, cavity morphology, lift, drag etc.) are consistent to those obtained by alternate studies (Kunz et al. 2003, Coutier-Delgosha et al. 2003, Qin et al. 2003). Further examination in the context of filter-based modeling and cavitating flows was performed over the Clark-Y aerofoil and a convergent-divergent nozzle (Wu et al. 2004, 2005). The filter-based model produced pronounced time-dependent behavior in either case due to significantly low levels of eddy viscosity. While the time-averaged results showed consistency to experimental data, they were unable to capture the essence of unsteady phenomena in the flow-field such as wave propagation. In addition to implementing the two-equation model for turbulent viscosity, Athavale et al. (2000) also accounted for turbulent pressure fluctuations. Thus, the threshold cavitation pressure (\( P_v \)) was modified as:

\[
p'_v = p_v + p_i / 2
\]

(2.15)

The turbulent pressure fluctuations were modeled as follows:

\[
p_i = 0.39 \rho_m k
\]

(2.16)

Though their computations produced consistent results, the precise effect of incorporating the turbulent pressure fluctuations was not discerned.
In addition to the above review, Wang et al. (2001), Senocak and Shyy (2002, 2004a, 2004b), Ahuja et al. (2001), Venkateswaran et al. (2002), and Preston et al. (2001) have also reviewed the recent efforts made in computational and modeling aspects.

2.2 Modeling Thermal Effects of Cavitation

Majority of studies on cavitation have made the assumption of isothermal conditions since they focused on water. However, as explained earlier, these assumptions are not suitable under cryogenic conditions because of their low liquid-vapor density ratios, low thermal conductivities, and steep slope of pressure-temperature saturation curves. Efforts on experimental and numerical investigation of cryogenic cavitation are dated as back as 1969. Though the number of experimental studies on this front is restricted due to the low temperature conditions, sufficient benchmark data for the purpose of numerical validation is available. However, there is a dearth of robust numerical techniques to tackle this problem numerically. The following sub-sections provide fundamental insight into the phenomenon in addition to a literature review.

2.2.1 Scaling Laws

Similarity of cavitation dynamics is dictated primarily by the cavitation number ($\sigma$) defined as (Brennen 1994, 1995):

$$
\sigma = \frac{p_\infty - p_v(T_\infty)}{0.5 \rho U_\infty^2}
$$

(2.17)

Under cryogenic conditions, however, cavitation occurs at the local vapor pressure dominated by the temperature depression. Thus, the cavitation number for cryogenic fluids is modified as:

$$
\sigma_c = \frac{p_\infty - p_v(T_\infty)}{0.5 \rho U_\infty^2}
$$

(2.18)
where, $T_c$ is the local temperature in the cavity. The two cavitation numbers can be related by a first-order approximation as follows:

$$\frac{1}{2} \rho_f U_w^2 (\sigma_c - \sigma) = \frac{dp}{dT} (T_c - T_\infty)$$  \hspace{1cm} (2.19)

Clearly, the local temperature depression ($T_c - T_\infty$) causes an increase in the effective cavitation number, consequently reducing the cavitation intensity. Furthermore, equation (2.19) underscores the effect of the steep pressure-temperature curves shown in Chapter 1.

Quantification of the temperature drop in cryogenic cavitation has been traditionally assessed in terms of a non-dimensional temperature drop termed as $B$-factor (Ruggeri and Moore 1969). A simple heat balance between the two phases can estimate the scale of temperature difference caused by thermal effect.

$$\rho_v v_v L = \rho_l v_l C_{pr} \Delta T$$  \hspace{1cm} (2.20)

Here, $v_v$ and $v_l$ are volume flow rates for the vapor and liquid phase respectively. The $B$-factor can then be estimated as:

$$B = \frac{v_v}{v_l} = \frac{\Delta T}{\Delta T^*} ; \Delta T^* = \frac{\rho_l L}{\rho_l C_{pr}}$$  \hspace{1cm} (2.21)

Consider the following two flow scenarios for estimating $B$, as shown in Figure 6 (adapted from Franc et al. 2003).
Figure 6. Two cavitation cases for $B$-factor analysis

The estimation of $B$-factor for the case (a) (two-phase cavity) in above figure is expressed in following equation.

$$v_i \sim \alpha_i U_\infty \delta_c; \Delta v_\infty \sim (1-\alpha_i)U_\infty \delta_c; B = \frac{1-\alpha_i}{\alpha_i}$$  \hspace{1cm} (2.22)

This points the fact that except for the pure vapor region $B$ has an $O(1)$ value. For the case (b) in Figure 6 (adapted from Franc et al. 2003), where the cavity is assumed to be filled with 100% vapor, Fruman et al. (1991) provided an estimate of $B$ based on thermal boundary layer effect as:

$$B = \frac{\delta_c}{\delta_{\tau}} = \frac{\delta_c}{aL_c \sqrt{U_\infty}}$$  \hspace{1cm} (2.23)

Here, $a$ is the thermal or eddy diffusivity, $L_c$ is the cavity length, and $\delta_c$ and $\delta_{\tau}$ represent the thickness of the cavity and the thermal boundary layer, respectively. It is evident from equation (2.23) that the temperature depression is also strongly dependent on thermal diffusivity and flow properties. The temperature scale ($\Delta T^*$) in case of water and LH2 has a value of 0.01 and 1.2 K, respectively (Franc et al. 2003). The difference in these values provides an assessment of the pronounced thermal effects in LH2. Thus, by
knowing the values of $\Delta T$ and $B$ (from equations (2.22) and (2.23)), the actual temperature drop can be estimated. The $B$-factor, however, fails to consider time-dependent or transient thermal effects due to its dependence on a steady heat balance equation. Furthermore, the sensitivity of vapor pressure to the temperature drop, which is largely responsible in altering cavity morphology (Deshpande et al. 1997), is not accounted by it. As a result, though the $B$-factor may estimate the temperature drop reasonably, it is inadequate to evaluate the impact of the thermal drop on the cavity structure and the overall flow.

Brennen (1994, 1995) developed a more appropriate parameter to assess the thermodynamic effect by incorporating it into the Rayleigh-Plesset equation (equation(2.24)) for bubble dynamics.

$$\rho_l[R \frac{d^2 R}{dt^2} + \frac{3}{2} \left( \frac{dR}{dt} \right)^2] = p_v(T_v) - p_\infty$$

(2.24)

With help of equation(2.19), we can re-write above equation as:

$$\rho_l[R \frac{d^2 R}{dt^2} + \frac{3}{2} \left( \frac{dR}{dt} \right)^2] + \frac{dp_v}{dT} \Delta T = p_v(T_v) - p_\infty$$

(2.25)

From standpoint of a transiently evolving bubble, the heat flux $q$ at any time $t$ can be expressed as:

$$q = K \frac{\Delta T}{\sqrt{at}}$$

(2.26)

The denominator in equation(2.26), $\sqrt{at}$, represents the thickness of the evolving thermal boundary layer at time $t$. The heat balance across the bubble interface is expressed as follows.

$$q4\pi R^2 = \rho_v L \frac{4}{3} \pi R^3$$

(2.27)
Combining equations (2.26) and (2.27) we obtain:

\[
\Delta T \approx \frac{\hat{R} \sqrt{t}}{\sqrt{a}} \frac{\rho_i L}{\rho_l C_p} \tag{2.28}
\]

Introducing this temperature difference into equation (2.25) we obtain:

\[
[R \frac{d^2R}{dt^2} + \frac{3}{2} \left( \frac{dR}{dt} \right)^2] + \Sigma \hat{R} \sqrt{t} = \frac{p_v(T_v) - p_\infty}{\rho_i} \tag{2.29}
\]

A close observation yields the fact that the impact of thermal effect on bubble dynamics depends on \( \Sigma \) which is defined as:

\[
\Sigma = \frac{\rho_i L}{\rho_l C_p \sqrt{a}} \frac{dP_v}{d\hat{R}} \tag{2.30}
\]

The units of \( \Sigma \) are m/s^{3/2} and it proposes a criterion to determine if cavitation process is thermally controlled or not.

Franc et al. (2003) have recently extended the above analysis to pose a criterion for dynamic similarity between two thermally controlled cavitating flows. They replaced the time dependency in equation (2.29) by spatial-dependency through a simple transformation \( x = U_\infty t \). Here, \( x \) is the distance traversed by a bubble in the flow-field in time \( t \). If \( D \) is chosen as a characteristic length scale of the problem, the Rayleigh-Plesset equation can be recast in the following form:

\[
\left[ \frac{\hat{R} \ddot{R}}{2} + \frac{3}{2} \frac{\dot{R}^2}{R} \right] + \Sigma \frac{D}{U_\infty^3} \hat{R} \sqrt{\bar{F}} = -\frac{C_p + \sigma}{2} \tag{2.31}
\]

In equation (2.31), all the quantities with a bar are non-dimensional, and \( C_p \) is the pressure coefficient. All the derivatives are with respect to \( \bar{x} = x / D \). The above equation points out the fact that, besides \( \sigma_\infty \), two thermally dominated flows can be dynamically
similar if they have a consistent value of the non-dimensional quantity \( \Sigma \sqrt{D/U_\infty^3} \). It is important to note the important role of velocity scale \( U_\infty \) in the quantification of thermal effect at this juncture. Franc et al. (2003) suggested that though thorough scaling laws for thermosensible cavitation are difficult to develop, a rough assessment may be gained from above equation. However, it is imperative to highlight that all the above scaling laws have been developed using either steady-state heat balance or single bubble dynamics. As a consequence, their applicability to general engineering environments and complex flow cases is questionable.

The following sections will discuss the experimental and numerical investigations of cavitation with thermal consideration. Particularly, emphasis is laid on the limitations of currently known numerical techniques. Experimental studies are cited solely according to their relevance to the current study.

2.2.2 Experimental Studies

Sarosdy and Acosta (1961) detected significant difference between water cavitation and Freon cavitation. Their apparatus comprised a hydraulic loop with an investigative window. While water cavitation was clear and more intense, they reported that cavitation in Freon, under similar conditions, was frothy with greater entrainment rates and lower intensity. Though their observations clearly unveiled the dominance of thermal effects in Freon, they were not corroborated with physical understanding or numerical data. The thermodynamic effects in cavitation were experimentally quantified as early as 1969. Ruggeri et al. (1969) investigated methods to predict performance of pumps under cavitation conditions for different temperatures, fluids, and operating conditions. Typically, strategies to predict the Net Positive Suction Head (NPSH) were developed.
The slope of pressure-temperature saturation curve was approximated by the Clausius-Clapeyron equation. Pump performance under various flow conditions such as discharge coefficient and impeller frequency was assessed for variety of fluids such as water, LH2, and butane. Hord (1973a, 1973b) published comprehensive experimental data on cryogenic cavitation in ogives and hydrofoils. These geometries were mounted inside a tunnel with a glass window to capture visuals of the cavitation zone. Pressure and temperature were measured at five probe location over the geometries. Several experiments were performed under varying inlet conditions and their results were documented along with the instrumentation error. As a result, Hord’s data are considered benchmark results for validating numerical techniques for thermodynamic effects in cavitation.

From standpoint of latest investigations, Fruman et al. (1991) proposed that thermal effect of cavitation can be estimated by attributing a rough wall behavior to the cavity interface. Thus, heat transfer equations for a boundary layer flow over a flat plate were applied to the problem. The volume flow rate in the cavity was estimated by producing an air-ventilated cavity of similar shape and size. An intrinsic limitation of the above method is its applicability to only sheet-type cavitation. Larrarte et al. (1995) used high-speed photography and video imaging to observe natural as well as ventilated cavities on a hydrofoil. They also examined the effect of buoyancy on interfacial stability by conducting experiments at negative and positive AOA. They reported that vapor production rate for a growing cavity may differ substantially from the vapor production rate of a steady cavity. Furthermore, there may not be any vapor production during the detachment stage of the cavity. They also noticed that cavity interface under the effect of
gravity may demonstrate greater stability. Fruman et al. (1999) investigated cavitation in R-114 on a venturi section employing assumptions similar to their previous work (Fruman et al. 1991). The temperature on the cavity surface was estimated using the following flat plate equation.

\[ T_{plate} = T_\infty + \frac{q}{0.5 \rho_l C_p U_{x} C_f} \left[ 1 - \frac{2.1}{Re_{x}^{0.1}}(1 - Pr) \right] \]  

(2.32)

Note that \( C_f \) is the coefficient of friction and \( T_{plate} \) is assumed to be equal to the local cavity surface temperature. The heat flux \( q \) on cavity surface was estimated as:

\[ q = -\rho_l L U_{x} C_Q \]  

(2.33)

The discharge coefficient \( C_Q \) was obtained from an air-ventilated cavity of similar size. They estimated the temperature drop via the flat plate equation and measured it experimentally as well. These two corresponding results showed reasonable agreement.

Franc et al. (2001) employed pressure spectra to investigate R-114 cavitation on inducer blades. The impact of thermodynamic effect was examined at three reference fluid temperatures. They reported a delay in the onset of blade cavitation at higher reference fluid temperatures, which was attributed to suppression of cavitation by thermal effects.

Franc et al. (2003) further investigated thermal effects in a cavitating inducer. By employing pressure spectra they observed shift in nature of cavitation from alternate blade cavitation to rotating cavitation with decrease in cavitation number. The earlier is characterized by a frequency \( 2f_b \) (\( f_b \) is the rotor frequency with 4 blades) while the later is characterized by a resonant frequency \( f_b \). Furthermore, R-114 was employed as the test fluid with the view of extending its results to predicting cavitating in LH2. The scaling
analysis provided in section 2.2.1 was developed by Franc et al. (2003) mainly to ensure
dynamic similarity of cavitation in their experiments.

2.2.3 Numerical Modeling of Thermal Effects

Numerical modeling has been implemented in cavitation studies broadly for two
thermodynamic aspects. Firstly, attempts to model the compressible/pressure work in
bubble oscillations have been made. Lertnuwat et al. (2001) modeled bubble oscillations
by applying thermodynamic considerations to the Rayleigh-Plesset equation, and
compared the solutions to full DNS calculations. The bubble model showed good
agreement with the DNS results. The modeled behavior, however, deviated from the
DNS solutions under isothermal and adiabatic assumptions.

![Figure 7. Schematic of bubble model for extracting speed of sound](image)

Figure 7. Schematic of bubble model for extracting speed of sound
Rachid (2003) developed a theoretical model for accounting compressive effects of a
liquid-vapor mixture. The actual behavior of the mixture along with the dissipative
effects associated with phase transformation was found to lie between two limiting
reversible cases. One in which phase change occurs under equilibrium at a constant
pressure, and the other in which the vapor expands and contracts reversibly in the
mixture without undergoing phase change. Rapposelli and Agostino (2003) recently extracted the speed of sound for various fluids such as water, LOX, LH2 etc. employing a bubble model and rigorous thermodynamic relationships. The control volume \((V)\) of the bubble is illustrated in Figure 7 (adapted from Rapposelli and Agostino 2003), and can be expressed as:

\[
V = (1 - \varepsilon_l)V_l + (\varepsilon_l)V_l + (1 - \varepsilon_v)V_v + (\varepsilon_v)V_v
\]  

(2.34)

The model assumed that thermodynamic equilibrium between the two phases is only achieved amidst fractions \(\varepsilon_l\) and \(\varepsilon_v\) of the total volume. Subsequently, the remaining fractions of the two phases were assumed to behave isentropically. If \(m_l\) and \(m_v\) are masses associated with the respective phases, the differential volume change \(dV\) can be expressed as:

\[
\frac{dV}{V} = -(1 - \varepsilon_l)\frac{\alpha_i}{\rho_i} \left( \frac{d\rho}{dp} \right)_l - (1 - \varepsilon_v)\frac{\alpha_v}{\rho_v} \left( \frac{d\rho}{dp} \right)_v

\]

\[
-(\varepsilon_l)\frac{\alpha_i}{\rho_i} \left( \frac{d\rho}{dp} \right)_{sat} - (\varepsilon_v)\frac{\alpha_v}{\rho_v} \left( \frac{d\rho}{dp} \right)_{sat}

\]

\[
+dm\left( \frac{\alpha_v}{m_v} - \frac{\alpha_l}{m_l} \right)
\]

(2.35)

A close observation of above equation yields that the modeling extremities of \(\varepsilon_l, \varepsilon_v = 0\) and \(\varepsilon_l, \varepsilon_v = 1\) also correspond to the thermodynamic extremities mentioned in above-mentioned analyses by Lertnuwat et al. (2001) and Rachid (2003) (italicized in the above description). Finally, substitution of various thermodynamic relations into equation (2.35) yields a thermally consistent speed of sound in the medium. Rapposelli and Agostino (2003) reported that their developed model was able to capture most features of bubble dynamics reasonably well.
The second thermodynamic aspect of cavitation, which also forms the focal point of this study, is the effect of latent heat transfer. The number of numerical studies, at least in open literature, in this regard is highly restricted. Reboud et al. (1990) proposed a partial cavitation model for cryogenic cavitation. The model comprised three steps, which were closely adapted for sheet cavitation, in form of an iterative loop.

(a) Potential flow equations were utilized to compute the liquid flow field. The pressure distribution on the hydrofoil surface was imposed as a boundary condition based on the experimental data. Actually, this fact led to the model being called ‘partial’. The interface was tracked explicitly based on the local pressure. The wake was represented by imposing a reattachment law.

(b) The vapor flow inside the cavity was solved by parabolized Navier Stokes equations. The change in cavity thickness yielded the increase in vapor volume and thus the heat flux at each section.

(c) The temperature drop over the cavity was evaluated with the following equation:

\[ q = K_t \frac{\partial T}{\partial y} \mid_y \]  

(2.36)

The value of turbulent diffusivity \( K_t \) in the computations was arbitrarily chosen to yield best agreement to the experimental results, and \( q \) was calculated from step (b).

The iterative implementation of steps (a) – (c) yielded the appropriate cavity shape in conjunction with the thermal effect. Similar 3-step approach was adopted by Delannoy (1993) to numerically reproduce the test data with R-113 on a convergent-divergent tunnel section. The main drawback of both the above methods is their predictive capability is severely limited. This is mainly because these studies do not solve the
energy equation and depend greatly on simplistic assumptions for calculating heat transfer rates.

Deshpande et al. (1997) developed an improved methodology for cryogenic cavitation. A pre-conditioned density-based formulation was employed along with adequate modeling assumptions for the vapor flow inside the cavity and the boundary conditions for temperature. The interface was captured with explicit tracking strategies. The temperature equation was solved only in the liquid domain by applying Neumann boundary conditions on the cavity surface. The temperature gradient on the cavity surface was derived from a local heat balance similar to equation (2.36). The bulk velocity inside the vapor cavity was assumed equal to the free stream velocity. Tokumasu et al. (2002, 2003) effectively enhanced the model of Deshpande et al. (1997) by improving the modeling of vapor flow inside the cavity. Despite the improvements in the original approach (Deshpande et al. 1997), it is important to underscore the limitation that both the above studies did not solve the energy equation inside the cavity region.

Hosangadi and Ahuja (2003, 2005), and Hosangadi et al. (2003) recently reported numerical studies on cavitation using LOX, LH2, and liquid nitrogen. Their numerical approach was primarily density-based. Their pressure and temperature predictions over a hydrofoil geometry (Hord 1973a) showed inconsistent agreement with the experimental data, especially (Hord 1973a) at the cavity closure region. Furthermore, Hosangadi and Ahuja (2005), who employed the Merkle et al. (1998) model in their computations, suggested significantly lower values of the cavitation model parameters for the cryogenic cases as compared to their previous calibrations (Ahuja et al. 2001) for non-cryogenic fluids.
The above review (summarized in Table 4) points out the limited effort and the wide scope for improving numerical modeling of thermal effects in cryogenic cavitation. A broadly applicable and more robust numerical methodology is expected to be a significant asset to prediction and critical investigation of cryogenic cavitation.
### Table 4. Summary of studies on thermal effects in cavitation

<table>
<thead>
<tr>
<th>Author and Year</th>
<th>Method (Experimental/Numerical)</th>
<th>Main Findings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sarosdy and Acosta 1961</td>
<td>Experimental Cavitation in hydraulic loop</td>
<td>Freon cavitates with a suppressed intensity as compared to water</td>
</tr>
<tr>
<td>Ruggeri et al. 1969</td>
<td>Experimental Cavitation in pumps</td>
<td>Provided assessment of pump NPSH under cryogenic conditions</td>
</tr>
<tr>
<td>Hord 1973</td>
<td>Experimental</td>
<td>Published comprehensive test data on ogives and hydrofoil</td>
</tr>
<tr>
<td>Fruman et al. 1991</td>
<td>Experimental Natural and ventilated cavities</td>
<td>Estimated temperature drop using flat plate boundary layer equations</td>
</tr>
<tr>
<td>Larrarte et al. 1995</td>
<td>Experimental High speed photography</td>
<td>Vapor production for growing cavities in cryogenic fluids may not be estimated by ventilated cavities</td>
</tr>
<tr>
<td>Fruman et al. 1999</td>
<td>Experimental Venturi section and R-114</td>
<td>Flat plate equations to model sheet cavitation can give good results under certain restrictions</td>
</tr>
<tr>
<td>Franc et al. 2001</td>
<td>Experimental Cavitation in pump inducers</td>
<td>Reported delay in onset of cavitation due to thermal effect</td>
</tr>
<tr>
<td>Franc et al. 2003</td>
<td>Experimental Cavitation in pump inducers</td>
<td>Provided scaling analysis to ensure dynamic similarity</td>
</tr>
<tr>
<td>Reboud et al. 1990</td>
<td>Numerical Potential flow equations</td>
<td>Semi empirical numerical model</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Did not solve energy equation</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Suitable for sheet cavitation</td>
</tr>
<tr>
<td>Delannoy 1993</td>
<td>Numerical Potential flow equations</td>
<td>Semi-empirical numerical model</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Did not solve energy equation</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Suitable for sheet cavitation</td>
</tr>
<tr>
<td>Deshpande et al. 1997</td>
<td>Numerical Explicit interface tracking</td>
<td>Density based formulation</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Simplistic approximation for cavity vapor flow</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Did not solve energy equation in the vapor phase</td>
</tr>
<tr>
<td>Tokumasu et al. 2002, 2003</td>
<td>Similar to Deshpande et al. (1997)</td>
<td>Improved flow model for cavity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Applicable to only sheet cavitation</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Did not solve energy equation in the vapor phase</td>
</tr>
<tr>
<td>Lertnuwat et al. 2001</td>
<td>Numerical Model for bubble oscillations</td>
<td>Incorporated energy balance for bubble oscillations</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Good agreement with DNS simulation</td>
</tr>
<tr>
<td>Rachid 2003</td>
<td>Theoretical Compression model liquid vapor mixture</td>
<td>Dissipative effects in phase transformation intermediate between two extreme reversible thermodynamic phenomena</td>
</tr>
<tr>
<td>Rapposelli &amp; Agostino 2003</td>
<td>Numerical Model for bubble oscillations</td>
<td>Employed thermodynamic relations to extract speed of sound for various liquids</td>
</tr>
<tr>
<td>Hosangadi &amp; Ahuja 2003, 2005</td>
<td>Numerical Density-based approach</td>
<td>Solved energy equation in the entire domain with dynamic update of material properties</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Some inconsistency with experimental results noted</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Significant change in the cavitation model parameters between non-cryogenic and cryogenic conditions</td>
</tr>
</tbody>
</table>
CHAPTER 3
STEADY STATE COMPUTATIONS

This chapter firstly delineates the governing equations that are employed in obtaining steady-state solutions to various cases on cryogenic cavitation. Theoretical formulation/derivation and computational implementation of various models, namely cavitation, turbulence, compressibility, and thermal modeling aspects, are further highlighted. The boundary conditions and steady-state results yielded by the computational procedure are discussed in detail following the description of the basic framework.

3.1 Governing Equations

The set of governing equations for cryogenic cavitation under the single-fluid modeling strategy comprises the conservative form of the Favre-averaged Navier-Stokes equations, the enthalpy equation, the \( k - \varepsilon \) two-equation turbulence closure, and a transport equation for the liquid volume fraction. The mass-continuity, momentum, enthalpy, and cavitation model equations are given below:

\[
\frac{\partial \rho_m}{\partial t} + \frac{\partial (\rho_m u_j)}{\partial x_j} = 0 \tag{3.1}
\]

\[
\frac{\partial (\rho_m u_i)}{\partial t} + \frac{\partial (\rho_m u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu + \mu_t)(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij})\right] \tag{3.2}
\]

\[
\frac{\partial}{\partial t}[\rho_m (h + f_s L)] + \frac{\partial}{\partial x_j}[\rho_m u_j (h + f_s L)] = \frac{\partial}{\partial x_j}\left[(\frac{\mu}{Pr_L} + \frac{\mu_t}{Pr_t}) \frac{\partial h}{\partial x_j}\right] \tag{3.3}
\]
\[
\frac{\partial \alpha_i}{\partial t} + \frac{\partial (\alpha_i u_j)}{\partial x_j} = \dot{m}^+ + \dot{m}^-
\]  

(3.4)

We neglect the effects of compressible work and viscous dissipation from the energy equation because the temperature field in cryogenic cavitation is mainly dictated by the phenomenon of evaporative cooling. Here, the mixture density, sensible enthalpy, and the vapor mass fraction are respectively expressed as:

\[
\rho_m = \rho_i \alpha_i + \rho_v (1 - \alpha_i)
\]  

(3.5)

\[
h = C_p T
\]  

(3.6)

\[
f_v = \frac{\rho_i (1 - \alpha_i)}{\rho_m}
\]  

(3.7)

The general framework of the Navier-Stokes solver (Senocak and Shyy 2002, Thakur et al. 2002) employs a pressure-based algorithm and the finite-volume approach. The governing equations are solved on multi-block, structured, curvilinear grids. The viscous terms are discretized by second-order accurate central differencing while the convective terms are approximated by the second-order accurate Controlled Variations Scheme (CVS) (Shyy and Thakur 1994). The use of CVS scheme prevents oscillations under shock-like expansion caused by the evaporation source term in the cavitation model, while retaining second order of formal accuracy.

Steady-state computations are performed by discounting the time-derivative terms in the governing equations and relaxing each equation to ensure a stable convergence to a steady state. The pressure-velocity coupling is implemented through the SIMPLEC (Versteeg and Malalasekera 1995) type of algorithm, cast in a combined Cartesian-contravariant formulation (Thakur et al. 2002) for the dependent and flux variables, respectively, followed by adequate relaxation for each governing equation, to obtain
steady-state results. The physical properties are updated, as explained earlier, after every iteration.

3.1.1. Cavitation Modeling

Physically, the cavitation process is governed by thermodynamics and kinetics of the phase change process. The liquid-vapor conversion associated with the cavitation process is modeled through $m^+$ and $m^-$ terms in Eq. (3.4), which respectively represent, condensation and evaporation. The particular form of these phase transformation rates, which in case of cryogenic fluids also dictates the heat transfer process, forms the basis of the cavitation model. Given below are the three modeling approaches probed in the present study.

3.1.1.1 Merkle et al. Model

The liquid-vapor condensation rates for this particular model are given as (Merkle et al. 1998):

$$
\dot{m}^- = \frac{C_{dest} \rho_l \text{Min}(0, p - p_v) \alpha_l}{\rho_l (0.5 \rho_l U_\infty^2) t_\infty}
$$

$$
\dot{m}^+ = \frac{C_{prod} \text{Max}(0, p - p_v) (1 - \alpha_f)}{(0.5 \rho_l U_\infty^2) t_\infty}
$$

Here, $C_{dest} = 1.0$ and $C_{prod} = 80.0$ are empirical constants tuned by validating the numerical results with experimental data. The time scale ($t_\infty$) in the equation is defined as the ratio of the characteristic length scale to the reference velocity scale ($t_\infty = D / U_\infty$).

3.1.1.2 Sharp Interfacial Dynamics Model (IDM)

The source terms of this model are derived by applying mass and momentum balance across the cavity interface and appropriately eliminating the unquantifiable terms (Senocak and Shyy 2002, 2004a, 2004b).
A schematic of the cavity model is illustrated in Figure 8(a). The model relies on the assumption of a distinctly vaporous cavity with a thin biphasic zone separating it from the pure liquid region. The physical form of the source terms is given below:

\[
\begin{align*}
\dot{m}^- &= \frac{\rho_v \min(0, p - p_v) \alpha_i}{\rho_v (U_{v,n} - U_{l,n})^2 (\rho_l - \rho_v) \alpha_i} \\
\dot{m}^+ &= \frac{\max(0, p - p_v)(1 - \alpha_i)}{(U_{v,n} - U_{l,n})^2 (\rho_l - \rho_v) \alpha_i}
\end{align*}
\]

(3.9)

Here, the normal component of the velocity is calculated as (Senocak and Shyy 2002, 2004a):

\[
U_{v,n} = \bar{u} \cdot \bar{n}; \quad \bar{n} = \frac{\nabla \alpha_i}{|\nabla \alpha_i|}
\]

(3.10)

Due to implicit tracking of the interface, the interfacial velocity, \(U_{l,n}\), in unsteady computations needs modeling efforts. Previous studies simplistically expressed the interfacial velocity \(U_{l,n}\) in terms of the vapor normal velocity (Senocak and Shyy 2002, 2004a) \(U_{v,n}\). Alternate methods of modeling \(U_{l,n}\) are discussed in the Chapter 4 in the context of time-dependent simulations. In comparison, for the steady computations in this chapter, we impose \(U_{l,n} = 0\).
3.1.1.3 Mushy Interfacial Dynamics Model (IDM)

Experimental visualizations of cryogenic cavitation (Sarosdy and Acosta 1961; Hord 1973a, 1973b) have clearly indicated a mushy nature of the cavity. This salient characteristic of cryogenic cavitation solicits an adaptation of the existing cavitation model to reflect the same. We choose the Sharp IDM (Senocak and Shyy 2004a) in our analysis because of its stronger physical reasoning. The following discussion serves to revise the above cavitation model by re-examining its derivation and assumptions, to appropriately accommodate the features of cryogenic cavitation. Although we refer the reader to literature (Senocak and Shyy 2004a) for the detailed derivation of the earlier model, we underscore the differences between the two approaches in the proceeding description.

Figure 8(b) – representing Mushy IDM - depicts a cavity where the vapor pressure is a function of the local temperature, and the nature of cavitation demonstrates a weak intensity and consequently less probability for existence of pure vapor phase inside the cavitation zone. In comparison, Figure 8(a) – representing Sharp IDM - depicts a cavity typically produced under regular conditions (no temperature effects), characterized by a thin biphasic region separating the two phases. We initiate our approach similar to Senocak and Shyy (2004a) by formulating the mass and momentum balance condition at the cavity interface, which is assumed to separate the liquid and mixture regions. We neglect the viscous terms and surface tension effects from equation (3.12) assuming high Re flows and large cavity sizes respectively.

\[ \rho_l (U_{i,n} - U_{i,n}) = \rho_m (U_{m,n} - U_{i,n}) \]  
(3.11)

\[ P_l - P_v = \rho_m (U_{m,n} - V_{i,n})^2 - \rho_l (U_{i,n} - V_{i,n})^2 \]  
(3.12)
It is noteworthy that Senocak and Shyy (2004a) formulated the above equations midst the pure vapor and mixture regions while hypothesizing an interface between the two. The two models differ in terms of their interpretation of the cavity attributes, namely, the density and the velocity field within the cavitation zone, while adhering to the fundamental idea of mass/momentum balance. Since we attribute a “frothy” nature to the cavity, the term $\bar{u}\bar{n}$ is largely expected to represent the mixture normal velocity. Accounting for this fact, we eliminate the $U_{i,n}$ term from equation (3.12) using equation (3.11) as shown below.

$$p_l - p_v = \rho_m (U_{m,n} - U_{l,n})^2 - \rho_{m}^2 (U_{m,n} - U_{l,n})^2$$  \hspace{1cm} (3.13)

Further re-arrangement progressively yields the following equations:

$$p_l - p_v = \rho_m (U_{m,n} - U_{l,n})^2 [1 - \frac{\rho_{m}}{\rho_l}] = \rho_m (U_{m,n} - U_{l,n})^2 \frac{\rho_l - \rho_v}{\rho_l} \alpha_v$$  \hspace{1cm} (3.14)

$$\alpha_v = \frac{\rho_l (p_l - p_v)}{\rho_m (p_l - p_v)(U_{m,n} - U_{l,n})^2} = \frac{\rho_l (p_l - p_v)}{\rho_m (p_l - p_v)(U_{m,n} - U_{l,n})^2} (\alpha_v + \alpha_i)$$  \hspace{1cm} (3.15)

$$\alpha_v = \frac{\rho_l (p_l - p_v) \alpha_i}{\rho_m (p_l - p_v)(U_{m,n} - U_{l,n})^2} + \frac{\rho_l (p_l - p_v)(1 - \alpha_i)}{\rho_m (p_l - p_v)(U_{m,n} - U_{l,n})^2}$$  \hspace{1cm} (3.16)

From standpoint of formulating source terms for the $\alpha_i$ transport equation, we firstly normalize the above equation by the overall convective timescale ($t_\infty = D/U_\infty$). Secondly, we apply conditional statements on the pressure terms to invoke either evaporation or condensation depending on the local pressure and vapor pressure (refer to Senocak and Shyy 2002 & 2004a). Lastly, we assume that the volume rates for the individual phases are interchangeable barring the sign convention. For instance, the
evaporation term, $\dot{m}^-$, in the $\alpha_l$-equation or $\alpha_v$-equation would bear the same magnitude, but negative or positive sign, respectively. We thus obtain the following source terms from the above analysis:

$$\dot{m}^- = \frac{\rho_l \text{Min}(0, p - p_v)\alpha_l}{\rho_m (U_{m,n} - U_{l,n})^2 (\rho_l - \rho_v) \tau_w}$$

$$\dot{m}^+ = \frac{\rho_l \text{Max}(0, p - p_v)(1 - \alpha_l)}{\rho_m (U_{m,n} - U_{l,n})^2 (\rho_l - \rho_v) \tau_w}$$

(3.17)

The normal component of the mixture velocity is expressed as shown in equation (3.10) and consistent with the normal component of the vapor velocity employed by the Sharp IDM.

A comparison with equation (3.9) shows that a factor of $\rho_v/\rho_m$ weakens the evaporation term while a factor of $\rho_l/\rho_m$ strengthens the condensation term of the Sharp IDM to yield the Mushy IDM. For instance, at a nominal density ratio of 100 and $\alpha_l = 0.5$, the value of $\rho_v/\rho_m$ and $\rho_l/\rho_m$ is $1.82 \times 10^{-2}$ and 1.82, respectively. In terms of order of magnitude, the difference between the evaporation terms of the two models is relatively more pronounced than the condensation terms, especially for high liquid-vapor density ratios. Given this fact, we attempt to make the Mushy IDM consistent with the Sharp IDM by providing an exponential transition of the evaporation source term from one model to the other as a function of the phase fraction ($\alpha_l$). Specifically, we employ the evaporation term from equation (3.17) as we get closer to the liquid region, and that from equation (3.9) as we get farther from the liquid region. In comparison, the condensation term from equation (3.17), which is mainly expected to be active in the region outlining the ‘vaporous’ portion of the cavity, is unaltered. In summary, the Mushy IDM is eventually formulated as shown below.
\[
\begin{align*}
\dot{m}^- &= \frac{\rho_l \text{Min}(0, p - p_v) \alpha_i}{\rho_i (U_{m,n} - U_{i,n})^2 (\rho_i - \rho_v) \tau_x} \\
\dot{m}^+ &= \frac{\rho_l \text{Max}(0, p - p_v) (1 - \alpha_i)}{\rho_i (U_{m,n} - U_{i,n})^2 (\rho_i - \rho_v) \tau_x} \\
\frac{\rho_i}{\rho_-} &= \frac{\rho_i}{\rho_v} + (1.0 - \frac{\rho_i}{\rho_v}) e^{-\frac{1}{\beta} (1 - \alpha_i)} \\
\frac{\rho_i}{\rho_+} &= \frac{\rho_i}{\rho_m}
\end{align*}
\] (3.18)

Note that \( \beta \) is a free parameter, which regulates the switch between the evaporation terms and warrants calibration for different fluids. Its typical value could be \( O(0.1) \).

Figure 9. Behavior of \( \rho_i / \rho_- \) and \( \rho_i / \rho_+ \) vs. \( \alpha_i \) for the two models; \( \rho_i / \rho_v = 100 \) and \( \beta = 0.09 \).

Figure 9 contrasts the density-ratio terms (\( \rho_i / \rho_- \) and \( \rho_i / \rho_+ \)) of the Sharp IDM and Mushy IDM for the chosen parameter values.

### 3.1.2 Turbulence Modeling

For the system closure, the original two-equation turbulence model with wall functions is presented as follows (Thakur et al. 2002):

\[
\frac{\partial (\rho_m k)}{\partial t} + \frac{\partial (\rho_m u_j k)}{\partial x_j} = P_i - \rho_m \varepsilon + \frac{\partial}{\partial x_j} [\left( \mu + \frac{\mu_k}{\sigma_k} \right) \frac{\partial k}{\partial x_j}] 
\] (3.19)
\[
\frac{\partial (\rho_m \varepsilon)}{\partial t} + \frac{\partial (\rho_m u_i \varepsilon)}{\partial x_j} = C_{\varepsilon_1} \frac{\varepsilon}{k} P_t - C_{\varepsilon_2} \rho_m \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]
\] (3.20)

The turbulence production \(P_t\) and the Reynolds stress tensor is defined as:

\[
P_t = \tau_{ij} \frac{\partial u_i}{\partial x_j}; \quad \tau_{ij} = -\rho_m u_i' u_j'
\]

(3.21)

The parameters for this model, namely, \(C_{\varepsilon_1} = 1.44, C_{\varepsilon_2} = 1.92, \sigma_\varepsilon = 1.3, \sigma_k = 1.0\) are adopted from the equilibrium shear flow calibration (Launder and Spalding 1974). The turbulent viscosity is defined as:

\[
\mu_t = \frac{\rho_m C_{\mu} k^2}{\varepsilon}; \quad C_{\mu} = 0.09
\]

(3.22)

It should be noted that the turbulence closure and the eddy viscosity levels can affect the outcome of the simulated cavitation dynamics especially in case of unsteady simulations (as reviewed in Chapter 2). In this aspect, parallel efforts are being made in the context of filter-based turbulence modeling (Johansen et al. 2004; Wu et al. 2003c, 2004, 2005), which has shown to significantly increase the time-dependency in cavitating flows. We do not explore these techniques in the interest of steady-state simulations, which disregard time-dependent phenomena.

### 3.1.3 Speed of Sound (SoS) Modeling

Due to lack of dependable equation of state for liquid-vapor multiphase mixture, numerical modeling of sound propagation is still a topic of research. We refer the reader to past studies (Senocak and Shyy 2003, 2004a, 2004b, Wu et al. 2003b) for modeling
options, their impact and issues, and just outline the currently employed SoS model below.

\[ \text{SoS} = C_\rho = C(1 - \alpha_t) \]  \hspace{1cm} (3.23)

The density correction term in the continuity equation is thus coupled to the pressure correction term as shown below.

\[ \rho_m' = C_p p' \]  \hspace{1cm} (3.24)

Senocak and Shyy (2002 & 2004a) suggested an \( O(1) \) value for the constant \( C \) to expedite the convergence of the iterative computational algorithm. However, their recommendation is valid under normalized values for inlet velocity and liquid density. Since we employ dimensional form of equations for cryogenic fluids, we suggest an \( O(1/U_\infty^2) \) value for \( C \), which is consistent with the above suggestion in terms of the Mach number regime. The speed of sound affects the numerical calculation via the pressure correction equation by conditionally endowing it with a convective-diffusive form in the mixture region. In the pure liquid region, we recover the diffusive form of the pressure equation.

3.1.4 Thermal Modeling

The thermal effects are mainly regulated by the evaporative cooling process, which is further manifested by the temperature dependence of physical properties and vapor pressure.

3.1.4.1 Fluid property update

In the present study, we subject all the physical properties, namely, \( \rho_l, \rho_v, p_v, \mu, C_p, K, \) and \( L \) to temperature dependence.
Figure 10. Pressure-density and pressure-enthalpy diagrams for liquid nitrogen in the liquid-vapor saturation regime (Lemmon et al. 2002). Lines denote isotherms in Kelvin.

As indicated by Figure 10, the physical properties are much stronger functions of temperature than pressure, and can fairly assume the respective values on the liquid-vapor saturation curve at a given temperature. We update these properties from a NIST database (Lemmon et al. 2002) at the end of a computational iteration. Thus, we generate a look-up table of physical properties for a particular temperature range as a pre-processing step. Subsequently, for any temperature-based update, the table is searched by an efficient bisection algorithm (Press et al. 1992) and the required physical property is obtained by interpolating between the appropriate tabular entries.

3.1.4.2 Evaporative cooling effects

The energy equation, (Eq.(3.3)), is recast into the following temperature-based form, by separating the latent heat terms onto the right-hand-side.

\[
\frac{\partial}{\partial t} [\rho_m C_p T] + \frac{\partial}{\partial x_j} [\rho_m u_j C_p T] = \frac{\partial}{\partial x_j} [C_p (\frac{\mu}{Pr_x}) + \frac{\mu}{Pr_y} \frac{\partial T}{\partial x_j}] - \frac{\partial}{\partial x_j} [\rho_m (f, L)] + \frac{\partial}{\partial x_j} [\rho_m u_j (f, L)]
\]

(3.25)
As seen from equation (3.25), the ‘lumped’ latent heat terms manifest as a non-linear source term into the energy equation and physically represent the latent heat transfer rate. The spatial variation of thermodynamic properties and the evaporative cooling effect are intrinsically embedded into this transport-based source term. We calculate the source term by discretizing the associated derivatives in concert with the numerical schemes applied to the terms on the left-hand-side of the equation.

### 3.1.5 Boundary Conditions

The boundary conditions are implemented by stipulating the values of the velocity components (obtained from the experimental data), phase fraction, temperature, and turbulence quantities at the inlet. Furthermore, at the walls, pressure, phase fraction, and turbulence quantities are extrapolated, along with applying the no-slip and adiabatic condition on the velocity and temperature, respectively. Pressure and other variables are extrapolated at the outlet boundaries, while enforcing global mass conservation by rectifying of the outlet velocity components. In addition, we also hold the pressure at the reference pressure point constantly at the reference value (specified by the experiments). This is simply done by adjusting the linear coefficients of the pressure correction equation at that point, to yield zero correction, at every iteration. Symbolically, this is achieved by substituting $A_p^p = 1$, $A_{nb}^p = B_p^p = 0$ into the following linear equation at the point of interest:

$$
A_p^p p_p \dot{p}_p = \sum A_{nb}^p p_{nb} + B_p^p
$$

(3.26)

We observe that this adjustment imparts robustness and stability to the computation.
3.2 Results and Discussion

In this section, we firstly observe the nuances of the Mushy IDM on non-cryogenic cases. The purpose is to distill the impact solely caused due to the source term change between the Sharp and Mushy IDM, and contrast it against commonly referred experimental and computational results. Later, we extend our computations to the variable temperature environment of cryogenic fluids. Specifically, we observe the effect of the temperature field on the nature of cavitation under similar conditions. Furthermore, we assess the Mushy IDM with available experimental data (pressure and temperature) and compare it with the alternative cavitation models. In the process of calibrating the cavitation models for cryogenic fluids, we perform a global sensitivity analysis to evaluate the sensitivity of the prediction to changes in material properties and model parameters. We offer discussion about optimizing the model performance based on our sensitivity study.

3.2.1 Cavitation in Non-cryogenic Fluids

As mentioned above, we register first the impact of the Mushy IDM on non-cryogenic cases. In this initial exercise, we consider the influence of the mushy formulation only to the liquid boundary layer encompassing the cavity. Symbolically, we implement the following formulation for the non-cryogenic cases.

\[
\begin{align*}
\dot{m}^- &= \frac{\rho_l \text{Min}(0, p - p_v)\alpha_i}{\rho_l (U_{m,n} - U_{l,n})^2 (\rho_l - \rho_v) t_n} \\
\dot{m}^+ &= \frac{\rho_i \text{Max}(0, p - p_v)(1 - \alpha_i)}{\rho_i (U_{m,n} - U_{l,n})^2 (\rho_l - \rho_v) t_n}
\end{align*}
\]

if \( \alpha_i \leq 0.99 \) \( \frac{\rho_i}{\rho_+} = \frac{\rho_i}{\rho_v} \) else \( \frac{\rho_i}{\rho_+} = \frac{\rho_i}{\rho_m} \)

if \( \alpha_i \leq 0.99 \) \( \frac{\rho_i}{\rho_+} = 1 \) else \( \frac{\rho_i}{\rho_+} = \frac{\rho_i}{\rho_m} \) (3.27)
To prevent sudden discontinuity in the volume transfer rates, we perform a geometric smoothing operation over the source terms. Two flow configurations are discussed here, namely, cavitating flow over a hemispherical projectile (time-averaged experimental data by Rouse and McNown (1948) at \( Re = 1.36 \times 10^5 \)) and cavitating flow over the NACA66MOD hydrofoil (time-averaged experimental data by Shen and Dimotakis (1989) at \( Re = 2 \times 10^6 \)).

Figure 11. Illustration of the computational domains for hemispherical projectile and NACA66MOD hydrofoil (non-cryogenic cases)
The computational domains for the two geometries are depicted in Figure 11. We consider two grids with \(158 \times 66\) and \(292 \times 121\) points for the hemispherical geometry, which is axisymmetric. In case of the NACA66MOD hydrofoil, we defer to the judgment of a previous study (Senocak and Shyy 2004a), and employ the finer grid from that study.

![Figure 12](image)

**Figure 12.** Pressure coefficients over the hemispherical body \((\sigma = 0.4)\); \(D\) is the diameter of the hemispherical projectile. (a) Impact of grid refinement for Mushy IDM (b) Comparison between pressure coefficients of different models on the coarse grid.

We investigate the sensitivity of Mushy IDM to grid refinement for the two hemispherical body grids through the surface pressure plots in Figure 12. Note that the refinement factor in the vicinity of cavitating region is roughly 2.5-3. Figure 12(a) indicates a noticeable, though modest, effect of the grid refinement on the surface pressure. This can be mainly attributed to the grid-dependent geometric smoothing operation, which smears the mushy formulation over a larger portion in the coarser grid. Nonetheless, both the grids produce solutions that match the experimental data reasonably well. Furthermore, it is important to mention that the near-wall nodes of the coarser grid lie in the log-layer of the turbulent boundary layer enabling appropriate use of the wall function (Versteeg and Malalasekera 1995), while the fine grid is over-refined.
for the wall spacings to assume values in the suitable range. As pointed out by Senocak and Shyy (2004a), in view of the wall function treatment (Thakur et al. 2002; Versteeg and Malalasekera 1995), the presence of vapor in the cavity may substantially reduce the appropriate range of the wall node spacings. As a consequence, we employ the coarser grid for further computations. Figure 12(b) contrasts the performance of the Mushy IDM with the Merkle et al. Model (1998) and Sharp IDM (2004a) at $\sigma = 0.4$. The pressure coefficient predicted by the three models illustrates noticeable variations in the condensation region of the cavity. Specifically, the Mushy IDM, as seen from Figure 12 (b), tends to produce a sharper recovery of the surface pressure in the cavity closure zone. Additionally, the Mushy IDM produces a small pressure dip (below the vapor pressure value) at the cavity outset due to the lower evaporation rate and higher condensation rate.

![Figure 13](image)

**Figure 13.** Cavity shapes and flow structure for different cavitation models on hemispherical projectile ($\sigma = 0.4$). (a) Merkle et al. Model (b) Sharp IDM (c) Mushy IDM
These features are reflected by the cavity sizes depicted in Figure 13. Note that the Mushy IDM not only shrinks the cavity length, in comparison with the Sharp IDM, but also impacts the flow structure in the recirculation zone of the cavity closure region. The above findings support our approach in providing an appropriate transition between the two models and a careful calibration to the parameter $\beta$
.

Figure 14. Pressure coefficient over the NACA66MOD hydrofoil at two different cavitation numbers; $D$ is hydrofoil chord length. (a) $\sigma = 0.91$ (b) $\sigma = 0.84$

We further simulate cavitating flow around the NACA66MOD hydrofoil at two cavitation numbers, namely, 0.84 and 0.91, while maintaining the mushy formulation only for the cavity boundary. Again, consistent behavior in terms of surface pressure is induced by the Mushy IDM for this geometry, at both the cavitation numbers (Figure 14). In summary, the appropriate source term modulation imposed by the Mushy IDM tends to influence the prediction of surface pressure and cavity size in manners consistent with the experimental observation. The above assessment motivates the further implementation and enhancement of the Mushy IDM (with respect to $\beta$) on cryogenic flow cases.
3.2.2 Cavitation in Cryogenic Fluids

In this section, we investigate the Mushy IDM for the cryogenic situation. We perform computations on two geometries experimentally investigated by Hord (1973a, 1973b), namely, a 2D quarter caliber hydrofoil (Hord 1973a) and an axisymmetric 0.357-inch ogive (Hord 1973b). These geometries were mounted inside suitably designed tunnels in the experimental setup.

Figure 15 illustrates the computational domains employed for these cases. Note that the figure shows only planar slices of the domains, which also model the respective tunnel shapes to account for the significant blockage effects. The mesh for the hydrofoil and the ogive geometry respectively comprises $320 \times 70$ and $340 \times 70$ points. The mesh distribution is chosen to facilitate adequate resolution of the cavitation zone.
Furthermore, the near-wall resolution over all the no-slip planes (cavitating geometries and tunnel walls) accounts for deployment of wall functions (Thakur et al. 2002; Versteeg and Malalasekera 1995).

Table 5. Flow cases chosen for the hydrofoil geometry.

<table>
<thead>
<tr>
<th>Fluid:</th>
<th>Case name:</th>
<th>Inlet temperature:</th>
<th>Freestream Re:</th>
<th>Cav. No. ($\sigma_*$):</th>
</tr>
</thead>
<tbody>
<tr>
<td>Liq. N$_2$</td>
<td>283B</td>
<td>77.65 K</td>
<td>4.7×10$^6$</td>
<td>1.73</td>
</tr>
<tr>
<td>Liq. N$_2$</td>
<td>290C</td>
<td>83.06 K</td>
<td>9.1×10$^6$</td>
<td>1.70</td>
</tr>
<tr>
<td>Liq. N$_2$</td>
<td>296B</td>
<td>88.54 K</td>
<td>1.1×10$^7$</td>
<td>1.61</td>
</tr>
<tr>
<td>Liq. H$_2$</td>
<td>248C</td>
<td>20.46 K</td>
<td>1.8×10$^7$</td>
<td>1.60</td>
</tr>
<tr>
<td>Liq. H$_2$</td>
<td>249D</td>
<td>20.70 K</td>
<td>2.0×10$^7$</td>
<td>1.57</td>
</tr>
<tr>
<td>Liq. H$_2$</td>
<td>255C</td>
<td>22.20 K</td>
<td>2.5×10$^7$</td>
<td>1.49</td>
</tr>
</tbody>
</table>

Source: Hord (1973a)

Table 6. Flow cases chosen for the ogive geometry.

<table>
<thead>
<tr>
<th>Fluid:</th>
<th>Case name:</th>
<th>Inlet temperature:</th>
<th>Freestream Re:</th>
<th>Cav. No. ($\sigma_*$):</th>
</tr>
</thead>
<tbody>
<tr>
<td>Liq. N$_2$</td>
<td>312D</td>
<td>83.00 K</td>
<td>9.0×10$^6$</td>
<td>0.46</td>
</tr>
<tr>
<td>Liq. N$_2$</td>
<td>322E</td>
<td>88.56 K</td>
<td>1.2×10$^7$</td>
<td>0.44</td>
</tr>
<tr>
<td>Liq. H$_2$</td>
<td>349B</td>
<td>21.33 K</td>
<td>2.3×10$^7$</td>
<td>0.38</td>
</tr>
</tbody>
</table>

Source: Hord (1973b)

Statistically-averaged pressure and temperature data are available for the two geometries at five probe locations over the body surfaces. The experimental findings report varying amounts of unsteady behavior in the cavity closure regions, although no case-specific information or data/visuals are available in that context. Hord conducted a series of experiments over both the geometries, using liquid nitrogen and hydrogen, by varying the inlet velocity, temperature, and pressure. In our computations, we have selected several
cases, referenced alphanumerically in the reports by Hord (1973a, 1973b), with different freestream temperatures and cavitation numbers (see Table 5 and Table 6).

Figure 16. Non-cavitating pressure distribution (a) case ‘290C’, D represents hydrofoil thickness and x represents distance from the circular bend (b) case ‘312D’, D represents ogive diameter and x represents distance from the leading edge

To validate the use of real fluid properties, we obtain the single-phase flow solution to cases ‘290C’ (hydrofoil; Re=9.1×10⁶, \( \sigma_\infty = 1.7 \)) and ‘312D’ (ogive; Re=9.0×10⁶, \( \sigma_\infty = 0.46 \)), and compare the computed surface pressure with the experimentally measured pressure under incipient (virtually non-cavitating) conditions. Figure 16 demonstrates good agreement between the numerical and experimental data for both the geometries, and corroborates the correct input of physical properties.

3.2.2.1 Sensitivity analyses

In general, cryogenic computations are prone to uncertainty due to a multitude of inputs, in contrast to the non-cryogenic conditions. The cavitation model parameters, namely, \( C_{dest}, C_{prod}, \) and \( t_\infty \), have been selected largely based on the non-cryogenic conditions. Furthermore, the solutions can exhibit substantial sensitivity with respect to minor changes in the flow environment. For example, the uncertainties involved in the
temperature-dependent material properties may also cause noticeable differences in predictions. To address relevant issues in this context and confirm/improve our calibration of the Merkle et al. model constants, we perform a comprehensive Global Sensitivity Analysis (GSA) over the chosen case ‘290C’. We initiate our computations on the case ‘290C’ (Re = 9.1×10^6; σ_∞ = 1.7) which is centrally located in the temperature range. We note that the previously calibrated values of the Merkle et al. Model (C_{dest} = 1.0 and C_{prod} = 80.0) are inadequate to provide a good match with the experimental data under the cryogenic condition. This fact was also lately noted by Hosangadi and Ahuja (2005), who suggested lower values of cavitation model parameters in case of cryogenic fluids.

Figure 17. Sensitivity of Merkle et al. Model prediction (surface pressure and temperature) to input parameters namely C_{dest} and C_{prod} for the hydrofoil geometry

Consistently, in the present study, we elicit C_{dest} = 0.68 and C_{prod} = 54.4 via numerical experimentation, as more appropriate model parameters. Of course, such choices are empirically supported and need to be evaluated more systemically.
To initiate such evaluations, Figure 17 portrays the response of the surface pressure and temperature to the revision of the model parameters. We choose $C_{\text{dest}}$, $t_{\infty}$, $\rho_{\infty}$, and $L$ as design variables for the GSA, while holding the Re and $\sigma_{\infty}$ constant for the given case. The chosen cavitation model parameters, namely $C_{\text{dest}}$ and $t_{\infty}$, are perturbed on either side of their reference values ($C_{\text{dest}} = 0.68$; $C_{\text{prod}} = 54.4$) by 15%. In comparison, the material properties are perturbed within 10% of the value they assume from the NIST database (Lemmon et al. 2002), at every iteration. Given these ranges on the variables, we generate a design of experiments with 50 cases using a combination of Orthogonal Arrays (Owen 1992) and Face Centered Cubic Design (JMP® 2002). RMS values of the hydrofoil surface pressure coefficient ($C_{p} = (p - p_{\infty})/(0.5 \rho_{\infty} U_{\infty}^{2})$) and temperature, which are post-processed from the CFD data, are selected as the objective functions. Subsequently, the two objectives are modeled by a reduced-quadratic response surface through a least-squares regression approach (JMP® 2002). The coefficient of multiple regression (Myers and Montgomery 1995) in case of the pressure and temperature fit is 0.992 and 0.993 respectively, while the standard error is less than 1%. The fidelity of the response surfaces is also confirmed against 4 test data points.

\begin{align}
C_{p_{\text{RMS}}} &= 1.675 + 0.077 C_{\text{dest}}^{*} - 0.061 \rho_{\nu}^{*} - 0.082 t_{\infty}^{*} - 0.007 C_{\text{dest}}^{*} \rho_{\nu}^{*} - 0.012 C_{\text{dest}}^{*} t_{\infty}^{*} + 0.011 \rho_{\nu}^{*2} + 0.009 \rho_{\nu}^{*} t_{\infty}^{*} - 0.004 L^{*2} + 0.013 t_{\infty}^{*2} \\
T_{\text{RMS}} &= 82.537 - 0.243 C_{\text{dest}}^{*} + 0.107 \rho_{\nu}^{*} - 0.048 L^{*} + 0.220 t_{\infty}^{*} + 0.024 C_{\text{dest}}^{*2} - 0.016 C_{\text{dest}}^{*} \rho_{\nu}^{*} - 0.015 C_{\text{dest}}^{*} L^{*} + 0.013 \rho_{\nu}^{*} L^{*} + 0.018 \rho_{\nu}^{*} t_{\infty}^{*} + 0.019 L^{*} t_{\infty}^{*} \nonumber \tag{3.29}
\end{align}

Here, $C_{p_{\text{RMS}}}$ and $T_{\text{RMS}}$ are the RMS values of the surface pressure coefficient and the surface temperature, respectively. The superscript, *, represents the normalized values of
the design variables. Equations (3.28) and (3.29) represent the respective response surfaces expressed in terms of the normalized design variables. The surface coefficients upfront indicate the importance of the cavitation model parameters \( C_{\text{dest}} \) and vapor density, and the insubstantial contribution of latent heat \( L \), to the variability in our objectives. We quantify these overall contributions by employing the variance-based, non-parametric global sensitivity method proposed by Sobol (1993). This method essentially comprises decomposition of the response surface into additive functions of increasing dimensionality. This allows the total variance in the data to be expressed as a combination of the main effect of each variable and its interactions with other variables (refer to Appendix A for a brief mathematical review).

![Figure 18. Main contribution of each design variable to the sensitivity of Merkle et al. (1998) model prediction; case ‘290C’](a) Surface pressure (b) Surface temperature)

We implement the procedure expounded by Sobol (1993) on the response surfaces in equations (3.28) and (3.29) to yield the plots in Figure 18. The pie-charts in the figure illustrate the percentage contribution of the main effect of each variable; since we find negligible variability due to the variable interactions. The charts firstly underscore the sensitivity of pressure and temperature predictions to the cavitation model parameters \( C_{\text{dest}} \) and \( t_\infty \). Secondly, the impact of \( \rho_v \) is noticeable, while that of \( L \) is
insubstantial. These observations indicate that the design variables, unlike $L$, which appear either in $\dot{m}^-$ or $\dot{m}^+$ may tend to register greater influence on the computed results. Thus, intuitively, $U_\infty$ and $\rho_l$, which are omitted from the present GSA, are expected to induce large variability in the computation, as compared to other omitted properties such as $K$ and $C_p$.

Figure 19. Pressure and temperature prediction for Merkle et al. Model for the case with best match with experimental pressure; $C_{dest}^* = 0.85; t_x^* = 0.85; \rho_v^* = 1.1; L^* = 0.9$

Figure 20. Pressure and temperature prediction for Merkle et al. Model for the case with best match with experimental temperature; $C_{dest}^* = 1.15; t_x^* = 0.85; \rho_v^* = 1.1; L^* = 1.1$
Furthermore, the impact is expected to be consistent on pressure and temperature, as depicted by Figure 18, due to the tight coupling between various flow variables. However, it is important to mention that our observation is meant to illustrate the relative impact of several parameters.

We also utilize the available data from our 50 cases to seek possible improvement of the Merkle et al. model parameters. Figure 19 illustrates the case which produces the least RMS error between the computed surface pressure and the experimental data. Conversely, Figure 20 portrays the case which produces the least RMS error between the computed surface temperature and the experimental data. These figures demonstrate that a single set of parameters may not provide optimal results for both pressure and temperature within the framework of current cavitation models and cryogenic conditions. This effort solicits multi-objective optimization strategies and deserves a separate study. However, we do report from the calculated error norms of the 50 cases that $C_{dest} = 0.68$ and $C_{prod} = 54.4$ provide the best balance between the temperature and pressure predictions for the chosen case. As a result, we hereafter employ these values for the Merkle et al. Model.

We perform a simpler sensitivity analysis over the Mushy IDM since it has only one control parameter ($\beta$).
Figure 21. Sensitivity of Mushy IDM prediction for case ‘290C’ (surface pressure and temperature) to the exponential transitioning parameter $\beta$.

Figure 21 depicts the results obtained over various values of $\beta$. We calibrate $\beta = 0.09$ from the observed results similarly based on a reasonable balance between pressure and temperature. Furthermore, Figure 21 suggests that the limiting case of $\beta = 0$, which essentially recovers the evaporation term of the Sharp IDM, would substantially over-predict the cavity size. This fact endorses the need to regulate the mass transfer rates modeled by the Sharp IDM, and subsequently the purported employment of the Mushy IDM. It is worthwhile to emphasize that the Mushy IDM manifests the regulation of mass transfer rates in cryogenic conditions – incorporated empirically into the Merkle et al. Model (section 3.2.1; Ahuja et al. 2001; Hosangadi and Ahuja 2005) - largely under physical pretext.

3.2.2.2 Assessment of cryogenic cavitation models over a wide range of conditions

We further perform computations for all the other cases by employing both the Merkle et al.’s and the present Mushy IDM cavitation models.
Figure 22. Surface pressure and temperature for 2-D hydrofoil for all cases involving liquid Nitrogen. The results referenced as ‘Mushy IDM’ and ‘Merkle et al. Model’ are contributions of the present study.
Figure 22 contrasts the resultant surface pressure and temperature obtained with the hydrofoil geometry for the cases involving liquid Nitrogen. We firstly note that both the models are able to provide a reasonable balance between pressure and temperature from standpoint of their predictive capabilities. The differences between the experimental data and the two models are more pronounced than the mutual differences between the two models. Furthermore, the agreement with experimental data is better in case of pressure than in case of temperature, which is generally under-predicted at the leading probe point by both the models. It is also observed that both models produce a slight temperature rise above the reference fluid temperature at the cavity rear end, which is attributed to the release of latent heat during the condensation process. The Merkle et al. Model also produces a steeper recovery of pressure, as compared to the Mushy IDM, in the condensation region of the cavity, for the cases shown. This suggests higher/faster condensation rates for the Merkle et al. Model than the Mushy IDM. Secondly, we assess our results for the case ‘290C’ along with latest computational data (Hosangadi and Ahuja 2005). Note that Hosangadi and Ahuja (2005) employed the Merkle et al. Model, adapted in terms of the vapor volume fraction ($\alpha_v$), with substantially higher values of the model coefficients ($C_{dest} = C_{prod} = 100$). The impact of these higher source term values is evident from the steep gradients observed in their surface temperature and surface pressure profiles. As a consequence, the temperature prediction of the present study appears better in the cavity closure region, while that of Hosangadi and Ahuja (2005) shows better agreement at the cavity leading edge. Thus, we emphasize again that the choice of model parameters poses a trade-off, as we noticed in the global sensitivity analysis, in the prediction of pressure and/or temperature for cryogenic cases. Lastly, it is
important to highlight that the inlet temperature gets closer to the critical temperature and the cavitation number decreases, as we proceed from case ‘283B’ to ‘296B’. In comparison, the inlet velocity and consequently the Reynolds number assume values within the same order of magnitude for the depicted cases. Thus, under isothermal conditions, an increase in the cavity length is expected from case ‘283B’ to ‘296B’. On the contrary, the surface pressure plots in Figure 22 clearly indicate a decrease in cavity length from case ‘283B’ to ‘296B’, despite the decrease in the freestream cavitation number. This fact clearly distills the significant impact of the thermal effect in cryogenic fluids, especially under working conditions that are close to the thermodynamic critical point.

![Figure 23](image)

**Figure 23.** Cavitation number \((\sigma = \frac{p_\infty - p_v(T)}{(0.5\rho_\infty U_\infty^2)})\) based on the local vapor pressure – Merkle et al. Model. Note the values of \(\sigma_\infty = \frac{p_\infty - p_v(T_\infty)}{(0.5\rho_\infty U_\infty^2)}\) for the cases ‘290C’ and ‘296B’ are 1.7 and 1.61, respectively.
Our contention on the thermal effect is corroborated by the cavitation number
\( \sigma = \left[ p_\infty - p_v(T_c) \right] / (0.5 \rho U_c^2) \) contours depicted in Figure 23. The freestream cavitation number \( \sigma_\infty \) of the case ‘296B’ is smaller than the case ‘290C’ (Table 5). However, the combination of evaporative cooling and its resultant impact over the vapor pressure causes a sharp increase in the effective cavitation number close to the cavitation zone. This increase is more substantial for the case ‘296B’ and eventually leads to comparable levels of effective cavitation number between the two cases, as seen in Figure 23.

![Cavity shape indicated by liquid phase fraction for case ‘290C’](image)

Figure 24. Cavity shape indicated by liquid phase fraction for case ‘290C’. Arrowed lines denote streamlines (a) Merkle et al. Model – isothermal assumption (b) Merkle et al. Model - with thermal effects (c) Mushy IDM – with thermal effects
Figure 24 reveals the phase fraction distribution and the streamlines for the computational cases of ‘290C’. The two models differ noticeably at the rear end of the cavitation zone. The cavity of the Mushy IDM consistently indicates lower condensation rates in contrast to the Merkle et al. Model, because of its longer length. We also note the gradual variation in density and the less extent of vapor phase in the cavities, which highlight the mushy/soggy nature of cavitation in cryogenic fluids. This is unlike our previous findings on regular fluids such as water (Senocak and Shyy 2004a, 2004b; Wu et al. 2003c). Under non-cryogenic conditions, the flow structure in the cavitation vicinity is generally characterized by large streamline curvatures and formation of recirculation zones (Senocak and Shyy 2002, 2004a, 2004b; Wu et al. 2003c) (also seen in Figure 13). However, the weak intensity of cavitation in cryogenic fluids has a modest impact over the flow structure, as indicated by the streamline patterns in Figure 24. Figure 24(a) illustrates a ‘special’ solution to the case ‘290C’ with the Merkle et al. Model assuming isothermal assumptions (energy equation not solved).

![Merkle et al. Model](image1)

![Mushy IDM](image2)

Figure 25 Evaporation ($\dot{m}^-$) and condensation ($\dot{m}^+$) source term contours between the two cavitation models – case ‘290C’. Refer to equations (3.8) and (3.18) for the formulations.
Note that discounting the thermodynamic behavior yields a substantially large cavity size under similar conditions. Based on these observations, we emphasize that, from a modeling standpoint, the thermal effect is manifested via a combination of the cavitation model adaptations and the temperature dependence of physical properties. We bolster our argument on the condensation rates between the two models in Figure 25. Note that the evaporation and condensation contour plots demonstrate a mutually exclusive behavior. The condensation region of the Merkle et al. Model illustrates sharper gradients and is effective over a much larger region, as compared to the Mushy IDM.

Following our assessment with liquid Nitrogen, we extend our focus to the cases with liquid Hydrogen, which has a density ratio of $O(30)$, unlike the $O(100)$ value for Nitrogen. We experience the need to re-calibrate our cavitation models for this different fluid because of the discernible role played by the density terms ($\rho_l$, $\rho_v$, and $\rho_m$) in determining the volume transfer rates. Our numerical experimentation yields $C_{\text{dest}} = 0.82$ and $C_{\text{prod}} = 54.4$ as appropriate values for the Merkle et al. Model, in case of liquid Hydrogen. Consistently, we choose $\beta = 0.065$ for the Mushy IDM in context of liquid Hydrogen. Our case selection for liquid Hydrogen, as seen from Table 6, follows similar trends as in case of liquid Nitrogen. The inlet velocity for all the cases with liquid Hydrogen is greater than 50 m/s, and it increases from the case ‘248C’ to ‘255C’ (66.4 m/s for ‘255C’). As a result, we upfront underscore these substantially higher values of inlet velocities that are employed for Hydrogen. Figure 26 depicts reasonable balance between the temperature and pressure predictions for the case ‘248C’; however, the agreement with the experimental data deteriorates equally for both the models as we
proceed from the case ‘248C’ to the case ‘255C’. Especially, the temperature is highly under-predicted as the inlet velocity increases between the two limiting cases.

Figure 26. Surface pressure and temperature for 2-D hydrofoil for cases involving liquid Hydrogen. The results referenced as ‘Mushy IDM’ and ‘Merkle et al. Model’ are contributions of the present study.
The results of Hosangadi and Ahuja (2005), while depicting the consistent aspect of sharp gradients that we noticed earlier, also portray significant discrepancies between the surface temperature and the experimental data at the rear region of the cavity. This finding could be either attributed to inadequacies in the cavitation model parameters or the representation of temperature-dependent physical properties. However, the observed sensitivity of predictions to physical properties, the relatively better agreement in the case ‘248C’, and the clear trend of disagreement with increasing velocities indicate the likelihood of the latter reason. Our extraction of physical properties from the NIST database (Lemmon et al. 2002) is based on models which assume thermodynamic equilibrium conditions. However, at such high fluid velocities, the timescales for thermodynamic equilibrium may be much larger than the overall flow timescales. This may introduce substantial discrepancies in the values of various physical properties and subsequently in the predictions; refer to Hosangadi and Ahuja (2005) for similar reporting. Of course, confirmation of the above possibility and development of rigorous non-equilibrium strategies is a challenging proposition for future research, and requires additional experimental insight. Nonetheless, we note the consistent performance of the Mushy IDM, especially in terms of the pressure prediction, over the chosen range of reference temperatures and velocities.

We finally attempt to re-assess our above calibrations \((\delta_{\text{dest}}, \delta_{\text{prod}}, \text{ and } \beta)\) for both the fluids and instill confidence into our predictive capabilities by performing computations over the axisymmetric 0.357-inch ogive geometry. The ogive surface pressure and temperature plots for all the three cases are displayed in Figure 27. The shrinkage of the cavity length between cases ‘312D’ and ‘322E’, despite the decrease in
the reference cavitation number, unequivocally re-emphasizes the influence of the thermo-sensible working conditions.

Figure 27. Surface pressure and temperature for axisymmetric ogive for all the cases (Nitrogen and Hydrogen). The results referenced as ‘Mushy IDM’ and ‘Merkle et al. Model’ are contributions of the present study.
Furthermore, the discrepancy in our predictions for liquid Hydrogen, when subjected to higher velocities, is also evidenced by the plots for the case ‘349B’. Overall, the agreement with experimental data in all the cases is better in terms of the surface pressure than temperature. But, unlike the hydrofoil geometry, the temperature at the first (leading) probe point is over-predicted for all the ogive cases. As expected, this discrepancy is most in the liquid Hydrogen case (‘349B’). Thus, the two geometries do not produce a consistent pattern of disagreement midst the surface temperature and the experimental measurements. This inconsistent behavior warrants further experimental and numerical probing from a standpoint of developing more precise cavitation models for cryogenic cavitation. Nonetheless, we observe that our analyses/calibrations are able to yield a justifiable range of results for the ogive geometry as well.
CHAPTER 4
TIME-DEPENDENT COMPUTATIONS FOR FLOWS INVOLVING PHASE CHANGE

Researchers have striven to develop efficient and accurate methodologies to simulate time-dependent cavitating flows. The multiphase nature of the flow accompanied by complex flow physics yields a system of tightly-coupled governing equations. Furthermore, interfacial dynamics, compressibility effects in mixture region, and turbulence entail deployment of numerical models to represent these physical phenomena. The formulation of these models substantially impacts the solution procedure. As a result, evolving efficient algorithms for unsteady cavitating flows is certainly a non-trivial task.

Formulation of implicit procedures for pressure-based methods is impeded mainly by the strong linkage between the flow variables such as velocity and pressure. Furthermore, the dynamics of the variable density field in cavitating flows imposes supplementary equations on the existing system, which add to the computational challenges. As a result, iterative algorithms such as SIMPLE, SIMPLER, and SIMPLEC (Versteeg and Malalasekera 1995), which are commonly employed for a wide variety of problems, may be computationally expensive for solution of cavitating flows with pronounced unsteady behavior. Senocak and Shyy (2002, 2004b) circumvented this difficulty by incorporating the Pressure Implicit with Splitting of Operators (PISO) algorithm suitably with the model equation of cavitation dynamics and the sound propagation model for the mixture region. This endeavor resulted in a non-iterative
methodology for time-dependent computation of cavitating flow through a series of predictor-corrector steps. Senocak and Shyy (2002, 2004b) demonstrated the merits of this efficient approach with a series of cavitating flow computations on geometries such as Convergent-Divergent Nozzle and hemispherical solids.

However, all the past efforts were in the context of isothermal cavitation *sans* the energy implications and real fluid properties. From standpoint of formulating a non-iterative algorithm for cryogenic cases, the inclusion of the non-linear energy equation and temperature dependence of physical properties into the existing methodology is expected to pose multitude of adversities to the solution efficiency and accuracy. Direct development of an algorithm for cryogenic cavitation may be a presumptuous preliminary approach. As a result, we initiate a multiphase non-iterative procedure on a simpler test problem with similar nature of challenges, and, with an objective of monitoring the accuracy and stability of the computations.

In this chapter, we elucidate the newly initiated algorithm and illustrate its results on a chosen test case. The truly unsteady problem of Gallium fusion (with natural convection effects) is adopted for the purpose of validation. Discussion on the grid sensitivity, accuracy of results, and the stability criterion is provided. The above primary objective is complimented by a reduced-order description of the gallium fusion problem by Proper Orthogonal Decomposition (POD). The flowfield in the problem is characterized by a solid-liquid front movement with a continuous change in the overall flow length scale. The ability of POD, to accommodate these varying flow scales, is mainly emphasized.
The insights gained from this study on gallium fusion are later extended to improving the algorithm for cryogenic cavitation. The algorithm in context of cryogenic cavitation is described to point out the changes implemented in order to address the temperature effects. POD is employed to probe the time-dependent data, and offer it a succinct representation.

4.1 Gallium Fusion

Experimental studies (Gau and Viskanta 1986) have investigated the physics of Gallium fusion due to its low fusion temperature and ease of handling. The availability of 2D experimental data has motivated numerical studies (Lacroix 1989, Lacroix and Voller 1990, Shyy et al. 1995) to adopt it as a test case.

![Schematic of the 2D Gallium square geometry with the Boundary Conditions (Shyy et al. 1998)](image)

Figure 28. Schematic of the 2D Gallium square geometry with the Boundary Conditions (Shyy et al. 1998)

A schematic of the square geometry along with the boundary conditions can be viewed in Figure 28.
4.1.1 Governing Equations

The single-fluid modeling approach is adopted by employing a liquid phase fraction \( f \). The density is assumed constant through the Boussinesq approximation. The governing equations for the problem are as follows.

\[
\frac{\partial u_i}{\partial x_i} = 0 \tag{4.1}
\]

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) - C \frac{1-f^2}{f^3+q} u_i + g \rho \beta (T - T_0) \delta_{ij} \tag{4.2}
\]

\[
\frac{\partial (\rho h)}{\partial t} + \frac{\partial (\rho u_i h)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( k \frac{\partial T}{\partial x_j} \right) \tag{4.3}
\]

Here, \( \beta \) represents the coefficient of thermal expansion for the fluid. The term \( C \frac{1-f^2}{f^3+q} u_i \) in the momentum equations is the Darcy source term (Shyy et al. 1998). Through their functional dependence on \( f \), they are modeled to retard the velocity to insignificant values in solid region. The constants \( C \) and \( q \), in compliance, are tuned to yield a negative source term, which is at least seven orders of magnitude higher than other terms, in the solid region (\( f = 0 \)). The enthalpy in equation (4.3) is expressed as:

\[
h = C_p T + f L \tag{4.4}
\]

The phase fraction \( f \) is modeled computationally by the \( h \)-based method (Shyy et al. 1998). The enthalpy at any given iteration is computed as shown below followed by an update of \( f \) (explained in a later section).

\[
h^k = C_p T^k + f^{k-1} L \tag{4.5}
\]

Note that \( f \) is always bounded by 0 and 1. Thus, equation (4.5) can be recast into a temperature equation as shown.
The iterative update of the liquid phase fraction, as mentioned above, renders the energy/temperature equation non-linear. Furthermore, the buoyancy term in equation (4.2) leads to a pressure-velocity-temperature coupling. The non-linearity in the energy equation and the strong coupling between various flow variables justify the use of the gallium fusion problem as a test case preceding the cryogenic problem. The physical properties such as viscosity, latent heat, thermal conductivity, and specific heat are, however, assumed to be constant during the fusion process.

### 4.1.2 Numerical Algorithm

The previous computations on similar multiphase problems employed an iterative solution strategy (Chuan et al. 1991, Khodadadi and Zhang 2001). Conversely, the PISO method (Issa 1985, Thakur et al. 2002, Thakur and Wright 2004), which forms the backbone of the current algorithm, essentially is a series of predictor corrector steps to yield a non-iterative solution of flow equations through operator splitting procedure. The present methodology, however, closely follows a slightly modified version of PISO designed for buoyancy driven single-phase flows (Oliveira and Issa 2001). Kim et al. (2000) proposed a series of steps to accelerate the simple heat conduction equation with a solid-liquid phase boundary. The following methodology attempts to blend the merits of the modified PISO (Oliveira and Issa 2001) with the propositions of Kim et al. (2000), to design an efficient and accurate algorithm for phase change problems. The sequence of calculations for the algorithm, based on the above governing equations, is expounded below.

\[
\frac{\partial}{\partial t}(\rho C_p T) + \frac{\partial}{\partial x_j}(\rho C_p u_j T) = \frac{\partial}{\partial x_j}(k \frac{\partial T}{\partial x_j}) - [\frac{\partial}{\partial t}(\rho L_f) + \frac{\partial}{\partial x_j}(\rho L u_j f)] \tag{4.6}
\]
The strongly implicit form of the discretized governing equations with a finite volume formulation, at any node $P$, is as follows:

$$
\Delta_t (\rho u^{n+1}_i) = 0
$$

$$
(A_p^u) u_p^{n+1} = \sum A_{nb}^u u_{nb}^{n+1} - \Delta_n p^{n+1} + H_p^u u_p^n + S_p^u (u^{n+1})
$$

$$
(A_p^v) v_p^{n+1} = \sum A_{nb}^v v_{nb}^{n+1} - \Delta_n p^{n+1} + H_p^v v_p^n + S_p^v (v^{n+1}) \quad (4.7)
$$

$$
(A_p^T) T_p^{n+1} = \sum A_{nb}^T T_{nb}^{n+1} + H_p^T T_p^n + H_p^f f_p^n + M (f^{n+1})
$$

Here, $A$ represents the terms at the current time level, $H$ represents the terms at older time step, $M$ represents the terms involving the phase fraction $f$, $B$ represents the buoyancy term, and $S$ represents the Darcy source term. The Darcy source terms can be absorbed into the left hand side term to yield to following equations.

$$
\Delta_t (\rho u^{n+1}_i) = 0
$$

$$
(G_p^u) u_p^{n+1} = \sum A_{nb}^u u_{nb}^{n+1} - \Delta_n p^{n+1} + H_p^u u_p^n
$$

$$
(G_p^v) v_p^{n+1} = \sum A_{nb}^v v_{nb}^{n+1} - \Delta_n p^{n+1} + H_p^v v_p^n + B_p^v (T^{n+1}) \quad (4.8)
$$

$$
(A_p^T) T_p^{n+1} = \sum A_{nb}^T T_{nb}^{n+1} + H_p^T T_p^n + H_p^f f_p^n + M (f^{n+1})
$$

Here, $G_p^u = A_p^u + S_p^u$ and so on. The sequence of steps to solve equation (4.8) non-iteratively is elaborated next.

(a) Momentum predictor

$$
(G_p^u) u_p^* = \sum A_{nb}^u u_{nb}^* - \Delta_n p^n + H_p^u u_p^n \quad (4.9)
$$

$$
(G_p^v) v_p^* = \sum A_{nb}^v v_{nb}^* - \Delta_n p^n + H_p^v v_p^n + B_p^v (T^n)
$$

These velocities - $u^*$ and $v^*$ - are obtained by using the pressure value at the previous time step and hence are not divergence free.
(b) First pressure corrector

\[ \Delta_i(\rho u_i^{**}) = 0 \]
\[ \Delta_i(\frac{\rho}{G_p} \Delta_i p^i) = \Delta_i(\rho u_i^i) \]  

(4.10)

Here, \( p^i = p^*_i - p \) is calculated. The pressure field \( p^*_i \) can now be employed for correcting the divergence error in velocity. Note that the pressure correction at this stage is limited by several approximations and is not adequate to produce an accurate velocity field.

(c) First momentum corrector

\[ (G_p^u)u_i^{**} = \sum A_{nb}^u u_{nb}^i - \Delta_e p^*_i + H_p^u u_p^i \]
\[ (G_p^v)v_i^{**} = \sum A_{nb}^v v_{nb}^i - \Delta_e p^*_i + H_p^v v_p^i + B_v(T^n) \]  

(4.11)

The velocities are corrected to yield \( u^{**} \) and \( v^{**} \) explicitly using the intermediate pressure field obtained in the previous step.

(d) First temperature corrector

\[ (A_p^T)T_p^{**} = \sum A_{nh}^T T_{nh}^n + H_p^T T_p^n + H_p^f f_p^n + M(f^*) \]  

(4.12)

The above temperature equation utilizes the latest value of \( f \). However, in view of a non-iterative strategy, solution of temperature, merely by the above equation, may not be sufficient for rapid convergence due to the delayed update of \( f \). As a remedy, a series of explicit steps are implemented following the above equation to significantly improve the prediction of temperature and, subsequently, \( f \). These steps are adapted from the algorithm proposed by Kim et al. (2000) for pure conduction equation. They are enlisted below.

(i) Update \( f \) by \( h \)-based method as shown.
\[ h_{p}^{k+1} = C_p T_p^{k+1} + f_p^k L \]
\[ f_p^{k+1} = 0 \text{ if } h_p^{k+1} < h_s \]
\[ f_p^{k+1} = 1 \text{ if } h_p^{k+1} > h_i \]
\[ f_p^{k+1} = \frac{h_p^{k+1} - h_s}{h_i - h_s} \text{ elsewhere} \]

where, \( h_s = C_p T_m, h_i = C_p T_m + L \) \hspace{1cm} (4.13)

Note that at the outset \( T_p^{k+1} = T_p^* \).

(ii) If \( f_p^{k+1} \neq f_p^k \), correct temperature using the explicit form of equation (4.12) with updated coefficients. This step yields the new temperature field \( T_p^{k+2} \).

(iii) Compute residual of the temperature equation as shown further.

\[ \Phi = (A^T_p) T_p^{k+2} - \sum A_{nb}^T T_{nb}^{k+2} + H^f T_p^n + H^f f_p^n + M (f^{k+1}) \] \hspace{1cm} (4.14)

The above residual equation can be further employed to improve temperature prediction by Newton-Raphson approach as follows (Kim et al. 2000):

\[ T_p^{k+3} = T_p^{k+2} - \frac{\Phi}{\partial T_p^{k+2}} \] \hspace{1cm} (4.15)

where, the Jacobian is approximated as \( \frac{\partial \Phi}{\partial T_p^{k+2}} \approx A^T_p \).

In summary, steps (i) – (iii) are performed at least 10-12 times following the equation (4.12). Let the temperature and phase fraction field at the end of this stage be denoted as \( T^* \) and \( f^* \).
(e) Second pressure corrector

\[ \Delta_l (\rho u_t^{**}) = 0 \]

\[ \Delta_l \left( \frac{\rho}{G_p^{**}} \Delta_p^{**} \right) = \Delta_l \left( \frac{\rho}{G_p^{**}} \left\{ B_p^r (T^r) - B_p^r (T^{**}) + \sum A_{nb}^{**} u^{**} - \sum A_{nb}^{**} u^r - \left[ (S_p^{i^r})^* - (S_p^{i}) u_p^{**} \right] \right\} \right) \]  

(4.16)

This step is a powerful characteristic of the PISO algorithm. The ‘differential Darcy term’, \((S_p^{i^r})^* - (S_p^{i}) u_p^{**}\), is an addition to the terms already suggested by Issa (1985) and Oliveira and Issa (2001). This term arises out of the update of \(G_p^{**}\) after the first temperature corrector. Note that the coefficient \(G_p^{**}\), unlike single-phase flow cases, also includes the implicit coefficient of the Darcy term (refer to the discussion in step (a)). The Darcy term undergoes a sudden change over several orders of magnitude between the liquid and the solid region. Since the first temperature corrector tends to change the phase fraction distribution, it is imperative to update the coefficient \(G_p^{**}\) post that step.

The second pressure corrector, which is obtained via subtracting equation (4.11) from (4.17), is thus able to account for phase front movement because of the ‘differential Darcy term’. In summary, the second pressure corrector attempts to couple the non-linear terms in the momentum equations, the temperature field, and the phase front movement to the pressure field.

(f) Second momentum corrector

\[ (G_p^r)^{**} u_p^{***} = \sum A_{nb}^{**} u_{nb}^{**} - \Delta_p^{**} p^{**} + H_p^{**} u_p^{**} \]

\[ (G_p^r)^{**} v_p^{***} = \sum A_{nb}^{**} v_{nb}^{**} - \Delta_p^{**} p^{**} + H_p^{**} v_p^{**} + B_p^r (T^r) \]  

(4.17)

The velocity field at this stage is derived from the more accurate prediction of pressure \(p^{**}\) obtained from the previous step.
Second temperature corrector

\[
(A_p^T)^* T_p^{**} = \sum A_{nb}^T T_{nb}^* + H_p^p T_p^n + H_p^f f^n + M^* (f^{**}) \quad (4.18)
\]

The prime purpose of this equation is to correct the temperature field with updated velocities. This step is followed by an update of \( f \), similar to that in equation (4.13).

It was reported that repeating the corrector steps (steps (b) – (g)) for an extra time improves coupling between various equations (Thakur and Wright 2004). However, due to the large magnitude of non-linearity expected for the fusion problem, at least 3 more repetitions of steps (b)-(g) are recommended (totally 8 corrector steps). The above modification to the PISO, similar to the original algorithm, incurs a splitting error due to the operator splitting approach. However, the order of this splitting error is higher than the formal order of temporal accuracy (Issa 1985), thus justifying the computational accuracy.

4.1.3 Results

The multiphase algorithm is implemented in conjunction with a pressure-based solver having multi-block capability (Thakur et al. 2002). The diffusive terms are handled by central differencing, while the first order upwind scheme is employed for the convective terms. Computations are mainly performed on a 2D square domain of size \( D=1 \). The range of values for \( St \) and \( Ra \) in the computations is \([1 – 0.042]\) and \([10^4 – 2.2\times10^6]\) respectively. Sample results obtained from current algorithm are compared to those obtained by Shyy et al. (1995) in the following discussion. It is important to note that results obtained by Shyy et al. (1995) have greater fidelity than those published in similar studies, due to the use of a finer grid.
4.1.3.1 Accuracy and grid dependence

Figure 29. 2D interface location at various instants for $St = 0.042$, $Ra = 2.2 \times 10^5$ and $Pr = 0.0208$. White circles represent interface locations obtained by Shyy et al. (1995) on a $41 \times 41$ grid at time instants at $t = 56.7s, 141.8s, & 227s$ respectively.

Figure 29 illustrates the 2D interface locations obtained with the current algorithm, with three different grid resolutions, for the parameters shown. Note that some results by Shyy et al. (1995) are at a slightly earlier time instant. This contributes to the modest difference observed in the interface location, especially during the early stages, when the interface velocity is significantly higher. Considering this fact, the present results show reasonable agreement with the earlier computation. Furthermore, the time-dependent movement of the interface is consistent on all the three grids, although there is a qualitative impact of the resolution on the interface profile.
Figure 30. Grid sensitivity for the $St = 0.042$, $Ra = 2.2 \times 10^5$ and $Pr = 0.0208$, 2D case (a) Centerline vertical velocity profiles at $t = 227s$ (b) Flow structure in the upper-left domain at $t = 57s$; $41 \times 41$ grid (c) Flow structure in the upper-left domain at $t = 57s$; $81 \times 81$ grid.

Table 7. Location of the primary vortex for the $St = 0.042$, $Ra = 2.2 \times 10^5$ and $Pr = 0.0208$ case

<table>
<thead>
<tr>
<th>Grid size</th>
<th>$x$-location at $t = 57s$</th>
<th>$y$-location at $t = 57s$</th>
<th>$x$-location at $t = 227s$</th>
<th>$y$-location at $t = 227s$</th>
</tr>
</thead>
<tbody>
<tr>
<td>21×21</td>
<td>0.130</td>
<td>0.760</td>
<td>0.350</td>
<td>0.674</td>
</tr>
<tr>
<td>41×41</td>
<td>0.135</td>
<td>0.755</td>
<td>0.365</td>
<td>0.661</td>
</tr>
<tr>
<td>81×81</td>
<td>0.146</td>
<td>0.710</td>
<td>0.377</td>
<td>0.658</td>
</tr>
</tbody>
</table>

The influence of the grid quality on the flow structure can be assessed from the centerline velocity profiles and the locations of the center of the primary vortex depicted in Figure 30(a) and Table 7 respectively. The even number of nodes (cell centers) in each grid creates a slight offset between the center gridline and the geometric centerline. This offset varies inversely with the grid quality, and is expected to contribute to the reasonable
discrepancy in the centerline velocity profiles, shown in Figure 30(a). In comparison, the
center of the primary vortex, from the findings reported in Table 7, indicates a gradual,
downward shift with grid refinement, particularly during the initial stages of fusion. In
fact, this movement, although moderate, is conspicuous at $t = 57s$, when an 81-point grid
is used instead of a 41-point grid. This observation has a physical relevance, as noticed
from Figure 30(b) and Figure 30(c). The upper-left part of the domain develops a
secondary vortex after some initial time lapse. This vortex is expected to induce a
downward motion of the primary vortex structure. As clearly seen, the $41 \times 41$ grid, unlike
the finer grid, is unable to capture this small-scale circulation, which justify the data in
Table 7. The overall flow convection pattern and, consequently, the interface movement
are, however, weakly affected by the secondary vortex structure. Furthermore, the
solution demonstrates a fairly consistent improvement with the grid quality, in addition to
a restrained grid-dependence. These findings are encouraging from the standpoint of
stability restrictions, which are elucidated in the following section.

Although the multiphase algorithm is elucidated/illustrated in context of 2D
calculations, its generality for 3D computations is examined by extending the case in
Figure 29 to square box geometry. The fusion process is initiated by two adjacent, heated
walls to yield spanwise flow variations.
Figure 31. 3D interface location at $t = 57s$ & $227s$ for $St = 0.042$, $Ra = 2.2 \times 10^5$ and $Pr = 0.0208$ case on a $41 \times 41 \times 41$ grid. Top and bottom: adiabatic; North and West: $T = 0$; South and East: $T = 1$ (heated walls)

Figure 32. Interface location and flow pattern for the 3D case, $St = 0.042$, $Ra = 2.2 \times 10^5$ and $Pr = 0.0208$ case at various $z$ locations, at $t = 227s$

Figure 31 and Figure 32 depict the interface movement and spanwise dependence of the flow structure respectively. Note that the time-step size for the 3D case is equal to that for $41 \times 41$ 2D grid. Accounting the dual heating surfaces, the results are qualitatively consistent to those in Figure 29 and instill confidence in 3D capability of the methodology.

4.1.3.2 Stability

As seen clearly, the basic framework of PISO incorporates several explicit corrector steps. As a result, the stable performance of the algorithm is restrained by an inevitable restriction on the time-step size. Numerical experimentation over the
previously-mentioned parametric range yielded the following heuristic stability criterion for the current problem (valid for 3D calculations).

\[
\text{CFL} = \frac{U \Delta \tau}{\Delta x} \leq 0.0018
\]

\[
\Delta \tau = \frac{\Delta t \alpha}{D \tau} St
\]

\[
U = \sqrt{g \beta \Delta T}
\]

(4.19)

All the results presented in this study employ the maximum permissible value of \(\Delta t\). Any grid refinement, as seen from above, will have a direct impact on the time-step size. However, the use of small time-step sizes is justifiable in case of truly unsteady flows in the interest of accuracy. Furthermore, the addition of extra corrector steps seems to increases the computational costs. But, it is worthwhile noting that these correctors steps are \(O(N)\) expensive, while the pressure solver is usually \(O(N^m)\) expensive (for instance, \(m = 1.5\) for Gauss-Seidel solvers), where \(N\) represents the total number of computational nodes. Thus, instead of employing iterative procedures and solving the expensive pressure equation numerous times, adding few smart \(O(N)\) corrector steps is much reasonable with the prospect of achieving the solution non-iteratively. In summary, the efficiency of the above algorithm is ensured by keeping the costs of the pressure solver low.

4.1.3.3 Data analysis by reduced-order description

Reduced-order examination of time-dependent flow problems by POD has been routinely pursued by researchers (Lumley 1967, Podvin 2001, Lucia et al. 2002, Ahlman et al. 2002, Zhang et al. 2003). A brief review on the latest developments and issues in the technique of POD is presented in Utturkar et al. (2005) and appendix B. POD essentially projects key features of a flowfield - derived from a data ensemble - in form of
orthogonal eigenmodes. Any flow variable at each instant can be expressed as a linear combination of the resultant eigenmodes as shown below:

\[ q(x, y, t) = \sum_{i=1}^{N} \phi_i(t) \psi_i(x, y) \]  \hspace{1cm} (4.20)

Here, \( \phi_i(t) = q^T(x, y, t) \psi_i(x, y) \) and \( \phi_i^2(t) \) is the amount of energy of \( \eta(x, y, t) \) in the ‘direction’ of \( \psi_i(x, y) \) (Sirovich 1987, Utturkar et al. 2005).

![Figure 33. POD modes showing velocity streamlines (\( \psi_i(r) \); \( i = 1, 2, 3, 4 \)) for \( St = 0.042, Ra = 2.2 \times 10^3 \) and \( Pr = 0.0208 \) case. \( q(r, t) = \tilde{V}(r, t) \).](image)

Useful information of the flowfield is optimally unfolded in order of the rapidly decreasing energy content of each mode. This facilitates accurate assessment of important issues through a reduced-order model by allowing truncation of equation (4.20) with fewer modes. Despite the proven capability of POD, in case of incompressible and compressible, and, laminar and turbulent flows, the present test case is expected to pose interesting issues to the technique. The flow field in the fusion problem is truly unsteady without a time-invariant mean component. There is a continuous growth in the flow...
domain and flow scale as the phase boundary traverses the domain. The ability of POD to accommodate these factors is the focal point of this brief investigation. Details on POD with regards to its mathematical background and numerical implementation can be found in Appendix B.

Figure 34 Scalar coefficients \( \phi_i(t); i = 1, 2, \ldots, 8 \) for \( St = 0.042, Ra = 2.2 \times 10^3 \) and \( Pr = 0.0208 \) case. \( q(r,t) = \bar{V}(r,t) \).

Figure 33 and Figure 34 illustrate the POD eigenmodes and scalar coefficients for the velocity field, for the flow case of \( St = 0.042, Ra = 2.2 \times 10^3 \) and \( Pr = 0.0208 \). 250 time-steps of the solution, which allow about 75% of the solid to fuse, are employed for the analysis. The energy fractions denote the cumulative kinetic energy recovered by the respective eigenmode. The first POD mode closely resembles the flowfield at the latest time instant. This mode, due to the truly unsteady behavior, is inconsistent with the time-averaged data, as in the previous cases, and just represents the flow structure with highest overall energy content. The following eigenmodes progressively represent the smaller structures in the flowfield. The structures shown by these successive modes are non-physical and are comparable to the harmonics in Fourier series decomposition. The first four modes of the chosen case recover almost 99% of the kinetic energy. However, as
explained further, this fact may be misleading to offer a concise representation under certain conditions.

The scalar coefficients (Figure 34) too show clear trends in their time-dependent behavior. They all initiate from zero and peak gradually in decreasing order of their ‘mode number’. Thus, the scalar coefficient of the first mode peaks latest. While the first peak in any coefficient denotes the dominance of the flow scales depicted by the respective eigenmode, the following peaks are relatively insignificant. This is because they are superseded by the coefficients of the preceding eigenmodes.

![Graph showing time instants when coefficients of respective POD modes show the first peak](image)

*Figure 35. Time instants when coefficients of respective POD modes show the first peak; \( St = 0.042, Ra = 2.2 \times 10^3 \) & \( Pr = 0.0208 \) case*

![Graph showing horizontal-centerline vertical velocities for different CFD solutions](image)

*Figure 36. Horizontal-centerline vertical velocities for \( St = 0.042, Ra = 2.2 \times 10^3 \) and \( Pr = 0.0208 \) case at \( t = 3s \)*
Figure 35 plots the ‘eigenmode number’ versus the time instance when the scalar coefficient shows its first peak. For instance, the second eigenmode peaks at $t = 20s$, whereas the twelfth mode peaks as early as $t = 3s$. Thus, at least two modes are required to represent the flow scales from any time $t > 20s$. In comparison, a reduced-order description for the flow from $t > 3s$ solicits at least twelve modes. This argument can be strengthened by observing the centerline vertical velocity profiles at time $t = 3s$, from Figure 36. The solid region, as seen from the plots, is manifested as a sharp discontinuity in the velocity field. Although the velocity profile using ten eigenmodes tends to represent the liquid region reasonably, it produces ripples in the solid region. The ripples are gradually attenuated by incorporating smaller scales of the higher-order POD modes; for instance, fifteen modes provide an appreciable representation of the profile at $t = 3s$ for both the phases, as seen from the figure. While the first twelve modes construct the flow scales in the liquid region, the additional modes suppress the oscillations in the solid zone.

![Comparison between CFD solution and re-constructed solution for $St = 0.042$, $Ra = 2.2 \times 10^3$ and $Pr = 0.0208$ case at $t = 20s$. Contours represent velocity magnitude and lines represent streamlines.](image)

Figure 37. Comparison between CFD solution and re-constructed solution for $St = 0.042$, $Ra = 2.2 \times 10^3$ and $Pr = 0.0208$ case at $t = 20s$. Contours represent velocity magnitude and lines represent streamlines.
Figure 38. Comparison between CFD solution and re-constructed solution for \(St = 0.042\), \(Ra = 2.2 \times 10^3\) and \(Pr = 0.0208\) case at \(t = 3\)s. Contours represent velocity magnitude and lines represent streamlines.

The above facts are corroborated by the CFD data and the re-constructed data, observed in Figure 37 and Figure 38, for two widely separated time instants. Impact of POD on the thermal field is not discussed here, because it is expected to be of similar nature. In general, the efficacy of a POD representation may strongly depend on the time interval of interest for truly unsteady problems, and may not merely rely on the energy content.

4.2 Turbulent Cavitating Flow under Cryogenic Conditions

The previous study pointed out the importance of extra corrector steps in case of strong non-linearity in an equation, and the requirement of additional source term(s) in the second pressure corrector in case of highly sensitive linear equation coefficients. These insights are applied to solving time-dependent cases of cryogenic cavitation in this section.

4.2.1 Governing Equations

The governing equations are exactly similar to those delineated in section (3.1). Some salient aspects of computational modeling involved in time-dependent simulations are discussed below.
4.2.1.1 Speed of sound modeling

While the speed of sound model in case of steady-state computations is chosen based on the criterion of fast convergence, for time-dependent computations we employ the following form (Senocak and Shyy 2002, 2004b; also reviewed in chapter 2).

\[
\text{SoS-2: } C_p = \left( \frac{\Delta p}{p_0} \right)_s = \left( \frac{\rho_0}{\rho} \right)_s = \frac{\rho_{i+1} - \rho_{i-1}}{p_{i+1} - p_{i-1}}
\]

(4.21)

The ability of the above formulation to produce a time-dependent behavior in close agreement with experimental observations has been previously demonstrated (Senocak and Shyy 2004b).

4.2.1.2 Turbulence modeling

Similar to the steady-state part, the two-equation based turbulence closure is adopted for time-dependent computations. In addition to the Launder and Spalding (1974) approach, we also employ the filter-based modeling (Johansen et al. 2004, Wu et al. 2004, Wu et al. 2005; also reviewed in chapter 2). The objective is to distill the impact of the turbulent viscosity field over the unsteady tendencies in the flow. For ease of reading, the filter-based formulation of eddy viscosity is again mention below.

\[
\mu_t = \frac{\mu_m C_k k^2}{\varepsilon} F, \quad C_k = 0.09
\]

(4.22)

where, the filter function, \( F \), is expressed in terms of a filter size (\( \Delta \)) as:

\[
F = \min[1, C_\Delta \frac{\Delta \varepsilon}{k^{3/2}}]; \quad C_\Delta = 1
\]

(4.23)

4.2.1.3 Interfacial velocity model

In the context of either Sharp IDM or Mushy IDM, the interfacial velocity (\( U_{i,n} \)) in time-dependent computations is non-zero, unlike steady-state cases, and needs modeling
efforts. Senocak and Shyy (2004b) simply correlated the interfacial velocity \( U_{I,n} \) to the vapor normal velocity \( U_{v,n} \) using the mass conservation formulation at the cavity interface. In the present study, we seek improvement to the modeling of \( U_{I,n} \). Consider the volume integral of the cavitation equation over the entire volume of the cavity, as shown:

\[
\int_{\text{cavity}} \frac{\partial \alpha_l}{\partial t} dV + \int_{\partial(\text{cavity})} [ (\alpha_l \vec{V}) \cdot \vec{n} ] da = \int_{\text{cavity}} (\dot{m}^+ + \dot{m}^-) dV
\]  

(4.24)

The second term on the left-hand-side represents the total flux of the liquid phase fraction over the cavity interface. In terms of the computational domain and a finite volume formulation, the above integral can be cast as:

\[
\sum_{\text{all cells}} \left( \frac{\partial \alpha_l}{\partial t} \times \text{cell volume} \right) + \sum_{\text{interface area}} \alpha_l(U_{v,n} - U_{I,n}) \times \text{interfacial area} = \sum_{\text{all cells}} [(\dot{m}^+ + \dot{m}^-) \times \text{cell volume}]
\]  

(4.25)

At this juncture, we defer to an assumption that the unsteadiness of \( \alpha_l \) is solely due to the movement of the interface. Thus, if there were no interface movement, we would get:

\[
\sum_{\text{interface area}} \alpha_l(U_{v,n}) \times \text{interfacial area} = \sum_{\text{all cells}} [(\dot{m}^+ + \dot{m}^-) \times \text{cell volume}]
\]  

(4.26)

Subtracting equation (4.26) from (4.25) and discounting the summation, we obtain a formulation for calculating the interfacial velocity at every node as follows:

\[
U_{I,n} = \frac{\frac{\partial \alpha_l}{\partial t} \times \text{cell volume}}{\alpha_l \times \text{interfacial area}}
\]  

(4.27)
The interfacial area at every cell is computed based on the areas of the cell faces and local normal direction \( \bar{n} = \bar{V} \alpha / |\bar{V} \alpha| \). Note that the above formulation yields \( U_{1,n} = 0 \) when \( \partial \alpha / \partial t = 0 \), and is thus consistent with steady-state behavior.

### 4.2.1.4 Boundary conditions

The boundary conditions for time-dependent cavitating computations are similar to those in the steady-state part barring the outlet condition. In case of unsteady calculations, the global mass conservation condition needs to account for the time-dependent mass accumulation inside the domain. This treatment may render the computation unstable, especially in context of the PISO algorithm. To circumvent this numerical difficulty, we impose a ‘mean’ (averaged over outlet plane) value for the pressure at the outlet plane in our simulations. Thus, the pressure correction is imposed zero at the outlet boundary. This condition is reasonable given the fact that the tunnels in the experimental setup (Hord 1973a, 1973b) were mounted between two cryogenic reservoirs (large Dewars).

### 4.2.2 Numerical Algorithm

The PISO algorithm is similarly adapted in case of cryogenic cavitation to handle the large density jumps (due to cavitation), the strong linkage between the temperature and \( \alpha \) equations, and the temperature-dependence of physical properties. The key steps of the algorithm in context of cryogenic cavitation are mentioned below. The changes made in the present context are specifically underscored while the identical steps are referred to the section (4.1.2).
The strongly implicit forms of the continuity, momentum, cavitation, and temperature equations are as follows (discretized forms of the $k$ & $\varepsilon$ equations are omitted due to their straightforward nature):

\[
\frac{\rho^{n+1} - \rho^n}{\Delta t} + \Delta_i (\rho u_i^{n+1}) = 0
\]

\[
(A_P^u)^{n+1} = \sum A_{mn}^u u_{nb}^{n+1} - \Delta_u p_{nb}^{n+1} + H_p^{u} u_P^{n+1}
\]

\[
(A_P^v)^{n+1} = \sum A_{mn}^v v_{nb}^{n+1} - \Delta_v p_{nb}^{n+1} + H_p^{v} v_P^{n+1}
\]

\[
(A_P^G + G_P^G)\alpha_P^{n+1} = \sum A_{mn}^G \alpha_{nb}^{n+1} + H_p^G \alpha_P^{n+1} + S_P^{G}
\]

\[
(A_P^T) T_P^{n+1} = \sum A_{mn}^T T_{nb}^{n+1} + H_p^T T_P^{n} + S_P^{T}
\]

Here, $S_p$ denotes the explicitly treated source terms in the cavitation and energy equation, while $G_p^G$ represents the implicitly treated source term of the cavitation equation.

(a) Momentum predictor

Logically identical to previous study

(b) First pressure corrector

This step is also similar to previous study except for the convective-diffusive form of the pressure corrector, as shown below ($C_p$ is modeled via the speed of sound).

\[
\frac{\rho^n - \rho^*}{\Delta t} + \Delta_i (\rho^n u_i^*) = \Delta_i (\rho^* u_i^*) + \Delta_i (\frac{C_p}{A_P^p} p^*)
\]

(c) First momentum corrector

Logically identical to previous study

(d) First scalar predictor
This step constitutes the prediction of $\alpha_l, T, k, and \varepsilon$. Note that the temperature and cavitation source terms are highly inter-linked. As a result, we resort to adding explicit corrector steps, as we implemented for the gallium problem. The phase fraction and temperature is firstly predicted as:

$$\begin{align*}
(A_p^\alpha + G_p^\alpha)\alpha_p^* &= \sum A_{nb}^\alpha\alpha_{nb}^* + H_p^\alpha\alpha_p^n + S_p^\alpha \\
(A_p^T)T_p^* &= \sum A_{nb}^T T_{nb}^* + H_p^T T_p^n + S_p^T
\end{align*}$$

(4.30)

Following this prediction, we perform the following series of steps 2-5 times.

(i) Update physical properties, namely, $\rho_l, \rho_v,$ and $p_v$ based on the predicted temperature ($T^*$).

(ii) Correct $\alpha_l$ explicitly by the pseudo Newton-Raphson technique, as per the previous study.

(iii) Correct temperature based on the corrected value of $\alpha_l$ obtained from (ii)

We note that as many as 5 repetitions are needed for cryogenic cases closer to the critical point, because of the steep variation of physical properties in that regime. However, 1-2 iterations of the above series may suffice for working conditions (temperature) away from the critical point.

(e) Update of physical properties

Values of all the physical properties, density, turbulent viscosity ($\mu_t$), and $C_p$ are updated at this point. Since cavitation is characterized by large changes in the density field, we also update the coefficients $A_p^\rho$ and $A_p^\nu$ in the momentum equations to account these changes.

(f) Second pressure corrector
\[
\frac{\rho^* - \rho^{**}}{\Delta t} + \Delta \left( \frac{\rho^*}{A^*_p} \Delta, p^{**} \right) = \Delta \left( \frac{\rho^*}{A^*_p} \left[ \sum A^*_w u^{**} - \sum A^*_w u^* \right] - [A^*_w, u^{**}] \right) + \Delta \left( \frac{C_u u^{**}}{A^*_p} - p^{**} \right) \quad (4.31)
\]

Due to the update on \( A^*_p \) and \( A^*_p \) in the previous step, the derivation of the second pressure corrector entails the inclusion of an extra term \( (A^*_w, u^{**} = (A^*_w - A^*_w) u^{**}) \) into its source terms. This term relates to the ‘differential Darcy term’ of the previous study, and couples the modifications/corrections in the density effects to the pressure field.

(g) Second momentum corrector

Logically identical to previous study

(h) First scalar corrector

Here, all the scalar equations (\( \alpha, T, k, \) and \( \varepsilon \)) undergo explicit corrections, as shown previously.

(i) Update of physical properties

Identical to step (e)

To enhance the coupling between various flow variables, we repeat all the above steps ((a) – (i)) one more time.

4.2.3 Results

Time-dependent experimental data in form of pressure signals and/or cavity snapshots are unavailable for cavitation in cryogenic fluids. As a consequence, we perform time-dependent computations over the 2D hydrofoil (Hord 1973a) that we employed for our steady-state cases. The details about the geometry and grid can be referred from chapter 3. Specifically, cases ‘283B’ and ‘290C’ are selected in conjunction with the filter-based model to preclude any undesirable dampening effects (filter size,
\[ \Delta = 0.11D, \ D \text{ is the chord length}. \] The time-step size is chosen in accordance with the CFL\(=U_\infty \Delta t / \Delta x \leq 1\) criterion.

\[
\begin{align*}
\text{(a)} & \quad \begin{array}{c}
\text{C}^p_t \\
\text{0.4} \\
\text{0.3} \\
\text{0.2} \\
\text{0.1} \\
\text{0} \\
\text{0.005} \\
\text{0.01} \\
\text{0.015} \\
\text{t(s)}
\end{array} \\
\text{(b)} & \quad \begin{array}{c}
\text{C}^p_t \\
\text{0.4} \\
\text{0.3} \\
\text{0.2} \\
\text{0.1} \\
\text{0} \\
\text{0.002} \\
\text{0.004} \\
\text{0.008} \\
\text{t(s)}
\end{array}
\end{align*}
\]

Figure 39. Pressure history at a point near inlet, \( \Delta = 0.11D \), where \( D \) is the chord length of the hydrofoil (a) ‘283B’ (b) ‘290C’

Figure 39 illustrates the time-dependent behavior of normalized pressure at a point near the tunnel inlet for the two cases. The plots indicate that, despite the deployment of filter-based turbulence model, the computation stabilizes to a steady state eventually.

\[
\begin{align*}
\alpha_l \\
\text{0.89} \\
\text{0.74} \\
\text{0.60} \\
\text{0.45} \\
\text{0.30}
\end{align*}
\]

Figure 40. Cavity shape and flow structure for case ‘283B’. Lines denote streamlines

\[
\begin{align*}
\alpha_l \\
\text{0.89} \\
\text{0.74} \\
\text{0.60} \\
\text{0.45} \\
\text{0.30}
\end{align*}
\]
The surface pressure and temperature obtained via the unsteady computations agree well with the steady state results in both cases (not shown here to avoid repetition of figures). The cause of this stable behavior can be explained based on the flow structure observed in Figure 40. Due to the remarkable weak intensity and mushy nature of cavitation in cryogenic fluids, the streamlines do not depict sharp deflections or formation of recirculation zones, unlike cavitation in regular fluids (Senocak and Shyy 2004a, 2004b; Wu et al. 2003c, 2004, 2005). In fact, the flow structure around the cavity does not seem to be noticeably different than the single-phase flow structure. As a consequence, the phenomenon of cavity auto-oscillations, which is mainly caused due to the re-entrant jet effects/instabilities (Senocak and Shyy 2004b), fails to occur in the above situation of cryogenic cavitation.

Figure 41. Pressure history at a point near the inlet for the case ‘290C’. St denotes the non-dimensional perturbation frequency ($fD/U_\infty = f_{\infty}$)

The time-dependent simulation strategy expounded in the previous section is further employed to investigate the impact of inlet temperature perturbation on the overall flow field. Unsteady computations are performed on cases ‘290C’ and ‘296B’, as
described in Chapter 3, by applying a sinusoidal boundary condition on the inlet
temperature. The amplitude of perturbation is chosen 0.5 K. A filter-size \( \Delta = 0.11D \) is
selected for the filter-based turbulence model.

![Figure 42](image_url)

Figure 42. Pressure history at a point near the inlet for the cases ‘296B’ and ‘290C’. The
non-dimensional perturbation frequency \( \left( \frac{fD}{U_\infty} = \frac{ft}{\infty} \right) \) is 1.0.

Figure 41 depicts the impact of the temperature perturbation on the pressure at an inlet
point for three different excitation frequencies. Note that the variation in vapor pressure
and phasic densities is about 5% given the temperature variation of 83.06 \( \pm \) 0.5 K. We
note that the amplitude of the pressure response is lowered at higher perturbation
frequencies. This mainly due to the timescale, \( t_\infty \), associated with the cavitation source
terms. At higher inlet frequencies, the response of the cavitation model to the rapidly
changing temperature environment is relatively slower. Due to the lack of this unison, we
observe a restrained oscillatory behavior at \( St = 2 \). Figure 42 illustrates the influence of
operating conditions on the tendency to respond to temperature fluctuations. The mean
temperature for case ‘290C’ and ‘296B’ is 83.06 K and 88.54 K, respectively. Since the
mean temperature in case ‘296B’ is closer to the critical point, the physical properties are
more temperature-sensitive as compared to the case ‘290C’. This fact is evident from Figure 42, which demonstrates larger amplitude of oscillation for the former case.

Figure 43 POD modes for the velocity field; \( \psi_i(r); i = 1, 2, 3; \quad q(r,t) = \tilde{V}(r,t) \)

Figure 44 POD modes for the velocity field; \( \psi_i(r); i = 1, 2, 3; \quad q(r,t) = \tilde{V}(r,t) \)
Since the degree of perturbation in the physical properties is about 5%, we fail to observe noticeable impact on the flow structure via instantaneous snapshots of the flow field. As a consequence, POD is employed to unearth the various levels of impact caused by the temperature perturbation on the flow structure. Figure 43 indicates that the first POD mode, which represents the mean/overall flow structure, is consistent between the two perturbation frequencies. However, we notice increasing amount of discrepancy between the corresponding plots of the higher order POD modes. Conversely, the POD modes in Figure 44, which contrasts cases ‘290C’ and ‘296B’, depict consistency in the flow structures of the two cases to a large extent. These trends can be explained by referring to Figure 41 and Figure 42. The dynamic behavior of the flow field under varying perturbation frequencies, as seen from Figure 41, shows different oscillatory amplitudes as well as dissimilar time-dependent patterns. In comparison, similar nature of perturbation in the two flow cases (‘290C’ and ‘296B’) seems to produce a major impact only on the amplitude of the response; the time-dependent pattern is consistent between the two flow configurations as observed from Figure 42. These facts are manifested likewise in the flow structure. Thus, the differences in flow cases ‘290C’ and ‘296B’ are largely absorbed by the time-dependent POD coefficients ($\phi_i(t)$) rendering consistent-looking modes of the flow structure ($\psi_i(r)$). In contrast, the effects of different perturbation frequencies are registered over the POD modes of the flow structure as well.
as the POD coefficients. The above observations may tend to vary at higher amplitudes of
the temperature perturbation, a phenomenon which may not be predicted upfront by the
above investigation. Nevertheless, the POD analysis serves to effectively discern the
effects of varying inlet temperature on the flow field by offering a succinct representation
to the time-dependent CFD data.
5.1. Summary

An effective computational procedure is formulated for simulating turbulent cavitating flow under cryogenic conditions. The existing framework of equations for turbulent cavitating flows is coupled with the energy equation to account for the evaporative cooling effect. The material properties are dynamically updated, as functions of temperature, to induce the thermo-sensible behavior into the cavitation characteristics. The derivation of the interfacial dynamics-based cavitation model is revised to accommodate the mushy features of cavitation typically observed for cryogenic fluids. The resultant model causes a non-linear decrease and increase in the evaporation and condensation source term of the original model, respectively.

The cavitation model parameters, which seem to be dependent on the fluid type, in the newly revised model (referred to as the Mushy IDM) and the Merkle et al. model are calibrated for liquid Nitrogen and Hydrogen via numerical experimentation. The sensitivity of predictions to these empirical choices and the uncertainties in material properties is systematically investigated through a global sensitivity analysis. The results indicate that the parameters and physical properties, which especially occur in the cavitation model source terms \( (C_{\text{det}}, C_{\text{prod}}, \beta, t, U, p, \rho, \rho_v, \rho_v, \text{and } \rho_\infty) \), are more likely to have a significant impact on the results than others \( (L, K, \mu, \text{and } C_p) \). Furthermore, the pressure and temperature fields pose a trade-off in terms of optimizing their prediction,
which is achieved by balancing their agreement with the experimental data. In comparison with the Merkle et al. Model, the Mushy IDM offers ease in calibration due to the presence of a single control parameter \( \beta \).

The performance and tendencies of the Mushy IDM depict consistency to the alternate models in context of the non-cryogenic cases. Subsequently, cryogenic computations are performed over two geometries (2D hydrofoil and axisymmetric ogive) with liquid Nitrogen and Hydrogen to extensively assess the framework over wide-ranging combinations of flow conditions and geometries. The thermodynamic effect is demonstrated under consistent conditions via the reduction in cavity length and the increase in effective cavitation number as the working conditions tend towards the critical point. Reasonable agreement between the computed surface pressure and temperature, and experimental data is obtained. Specifically, the agreement between the two models is closer than their respective agreement with the experimental data. In addition, the predictions of both the models are better; for the pressure field than the temperature field, and for liquid Nitrogen than liquid Hydrogen. The Hydrogen cases are characterized by very high inlet velocities, which seemingly have an adverse effect on the predictions. The deviation of temperature from the experimental values is not consistent between the hydrofoil and ogive geometry. This inconsistent behavior and the above-mentioned high velocity conditions for liquid Hydrogen warrant further experimental and numerical investigation for developing more precise models. Nevertheless, the Mushy IDM, whose derivation is based on the physical interpretation of cryogenic cavities, is able to offer reasonable results over all the flow cases.
A non-iterative algorithm for pressure-based calculation of multiphase flows is initiated from the viewpoint of simulating cryogenic cavitation. The Gallium fusion problem, which is characterized by a non-linear energy equation and strong pressure-velocity-temperature coupling, is employed as a test case. The algorithm adopts the framework of the operator splitting strategy (PISO) and incorporates steps to couple the phase front movement to velocity and pressure fields, and accelerate the convergence of the non-linear energy equation. Accurate and fairly grid-independent results over a wide-ranging parameter space are obtained. For inherently unsteady flow problems, the non-iterative formulation of the algorithm provides substantial computational benefit over the iterative approach, when accurate and time-dependent information is of interest.

The newly varied PISO algorithm is further adopted for time-dependent computations of cavitating flow in cryogenic fluids. Impact of perturbing the inlet temperature on the flow field is investigated via the time-dependent simulations. A stable performance of the algorithm is observed for all the cases. The weak nature of cavitation in cryogenic fluids has a modest influence over the flow structure, and thus tends to reduce the possibility of flow instabilities caused due to re-entrant jet effects. POD is employed to offer a succinct representation to the time-dependent data. POD is effectively able to distill dominant flow structures from the voluminous CFD data, and reveal the impact of the temperature perturbations and flow conditions on the unsteady behavior of the flow field.

5.2 Future Work

The present study may be extended in the following manner:

(a) Optimization of the cryogenic cavitation models via multi-objective strategies (namely Pareto Front analyses)
(b) Systematic experimental probing of the phase transformation rates involved in cavitation to enable development of more precise cavitation models

(c) Critical investigation of various assumptions in the case of liquid Hydrogen to reveal the cause of the observed discrepancies

(d) Investigation of compressibility and turbulence characteristics in cryogenic fluids

(e) Deployment/extension of the cryogenic cavitation framework in turbo-machinery applications
APPENDIX A
BACKGROUND OF GLOBAL SENSITIVITY ANALYSIS

The surrogate model (response surfaces in our case), $g(x)$, is decomposed as the sum of functions of increasing dimensionality as:

$$g(x) = g_0 + \sum_i g_i(x_i) + \sum_{i<j} g_{ij}(x_i, x_j) + \cdots + g_{12\ldots n}(x_1, x_2, \ldots, x_n)$$ (A.1)

Equation (A.1) is subjected to the following constraint:

$$\int_0^1 g_{h-1} dx_{h-1} = 0$$ (A.2)

Note that $x$ denotes normalized values of the design variables scaled between 0 and 1, and $g_0$ represents the mean value ($g_0 = \int g(x)dx$). The constraint in equation (A.2) renders the decomposition shown in equation (A.1) unique (Sobol 1993). Consequently, the total variance ($V$) in the data can be decomposed as:

$$V(y) = \sum_{i=1}^n V_i + \sum_{1 \leq i < j \leq n} V_{ij} + \cdots + V_{1\ldots n}$$ (A.3)

The partial variances, as shown in equation (A.3), are computed as follows:

$$V_i = V(E[y \mid x_i])$$
$$V_{ij} = V(E[y \mid x_i, x_j]) - V_i - V_j$$
$$V_{ijk} = V(E[y \mid x_i, x_j, x_k]) - V_i - V_j - V_k - V_{ij} - V_{ik} - V_{jk}$$ (A.4)

Here, $V$ and $E$ represent the variance and expected value respectively, and $i, j, k, s, n$ are the indices. Thus, the total contribution of a variable ($V_i^\text{total}$) to the variance in the data
can be expressed as the sum of the main effect of the variable and its interactions with other variables, as shown below.

\[ V_{i}^{\text{total}} = V_{i} + \sum_{j \neq i} V_{ij} + \sum_{j \neq i \neq k \neq i} V_{ijk} + \ldots \]  

(A.5)
APPENDIX B
REVIEW AND IMPLEMENTATION OF POD

B.1 Review

POD was first introduced by Lumley (1967) to investigate coherency of turbulent flow structures. Since this seminal contribution, POD has been a popular tool to extract systematically hidden, but deterministic, structures in turbulent flows and can be extensively found in literature. Gradual progress in this area has established the application of this technique to laminar flows, as well as to flows under incompressible and compressible conditions. The latest studies and relevant issues of POD implementation are reviewed hereafter. Aubry et al. (1988) built a five-mode, reduced-order model for the wall region of a fully turbulent channel. Their effort was extended by Podvin (2001), who provided numerical validation of a ten-dimensional model for the wall layer. The ability of the ten-dimensional model to produce intermittent features, which are reminiscent of bursting process in a wall layer, was also demonstrated. Comprehensive POD studies on turbulent mixing layers are available (Delville et al. 1999; Ukeiley et al. 2001), which not only identify the large-scaled structures in the layer, but also model the dynamic behavior of the lower-order modes. Prabhu et al. (2001) explored the effect of various flow control mechanisms in a turbulent channel, on the flow structure in POD modes. The POD modes of various flow control mechanisms showed significant differences close to the wall but were similar in the channel core. Similarly, while Liberzon et al. (2001) employed POD to study vorticity characterization in a turbulent boundary layer, Kostas et al. (2002) adopted the approach to probe PIV
data for the backward-facing step flow. Picard and Delville (2000) investigated the effect of longitudinal pressure distribution on velocity fluctuations in the turbulent shear layer of a subsonic round jet, using POD. Annaswamy et al. (2002) examined ‘edge-tones’ of an aircraft nozzle by analyzing the POD modes of azimuthal pressure distribution of a circular jet. Cizmas and Palacios (2003) gained insight on the turbine rotor-stator interaction with a lower-order POD investigation. They effectively utilized the time history and phase-plane plots of the POD coefficients to unravel the key dynamics in the flow behavior. POD-based investigations on pulsed jet flow field (Bera et al. 2001), temperature field in flow over heated waves (Gunther and Rohr 2002), and many such flow cases provide further evidence on the wide applicability of the technique. As mentioned earlier, POD has been successfully extended to laminar flow cases for extracting the principle features from time-dependent flow data (Ahlman et al. 2002).

Despite the ongoing progress in POD, several issues of its applicability to variable density and compressible flows are lately being examined. As indicated before, POD essentially yields a series, which rapidly converges towards the norm of a variable \(q(r,t)\). Several past studies intuitively adopted scalar-valued norms for the convergence criterion. For example, each flow variable namely pressure, density, or any velocity component was separately decomposed into POD modes. However, Lumley and Poje (1997) observed that for variable density flows, such as buoyancy-driven flows, an independent POD analysis may decouple the physical relationship between the flow variables. They suggested that the norm selection should incorporate the density variations into the velocity field, to achieve the convergence of a physically relevant quantity – mass rather than mere velocities - through the POD procedure. However,
simultaneous use of two flow variables also poses an important issue of deciding the significance of each variable in convergence process. Lumley and Poje (1997) suggested a vector form to \( q(r,t) \) as \( q(r,t)=[C_iu, C_i v, C_i w, C_i \rho'] \). In addition, they provided a mathematical analysis to optimize the values of the weighing factors for expediting the convergence. Colonius et al. (2002) extended the above argument by examining the impact of the norm selection on POD analysis of compressible flow over a cavity. POD modes independently obtained by scalar-valued norms of pressure and velocity were compared to those derived by vector-based norms. The choice of the vector, 
\[
q(r,t)=[u, v, w, \sqrt{\gamma - 1}c]
\]
was chosen with ingenuity so as to yield a norm as shown below:
\[
\| q(r,t) \| = (\int_{\Omega} (u^2 + v^2 + w^2 + \frac{2}{\gamma - 1}c^2) \, dr)^{\frac{1}{2}}
\]  
(B.1)

The above norm effectively yielded a linear series that converged towards the stagnation enthalpy instead of mere kinetic energy. Colonius et al. (2002) further reported that scalar-valued POD modes were unable to capture key processes such as acoustic radiation, which heavily rely on coupling mechanisms between the variables. In comparison, the vector-valued POD modes were in cohort with the compressible flow dynamics. Ukeiley et al. (2002) employed \( q(r,t)=[\rho, \frac{u}{U_{\infty}}, \frac{v}{U_{\infty}}, \frac{w}{U_{\infty}}, T - T_{\infty}] \) to perform POD on numerical data of compressible mixing layer. The variables, as shown above, were normalized by their freestream values to ensure a rational weighing of their fluctuations. A slow convergence towards the multi-variable norm was reported. POD implementation was also shown to face serious issues in case of supersonic flows due to
presence of shock fronts. Lucia et al. (2002) noticed that though the bulk flow can be modeled by few eigenmodes, a larger number of eigenmodes are required to accurately capture the discontinuity in the flow. They circumvented the issue by employing domain decomposition in their POD implementation. Though reasonable success in the reduced-order representation of the shock was reported, issues of extending the decomposition technique to moving shock fronts are still unresolved.

B.2 Mathematical Background

Consider a flow quantity, \( q(r,t) \), where \( r \) denotes spatial variables and \( t \) denotes time. The objective of POD can be simply stated as minimizing the \( L_2 \) norm of the objective error function, \( F_k(r,t) \), which is defined as (Lumley 1967; Delville et al. 1999; Ukeiley et al. 2001; Ahlman et al. 2002; Arian et al. 2002):

\[
F_k(r,t) = q(r,t) - q^k(r,t)
\]  

(B.2)

The \( L_2 \) norm in terms of volume integral can be defined as:

\[
\| F_k(r,t) \| = \left[ \int_{\Omega} F_k^2(r,t) dr^3 \right]^{1/2}
\]  

(B.3)

Here, \( q^k(r,t) \) denotes the data projected with a linear combination of certain number (\( k \)) of orthogonal basis functions (\( \psi(r) \)). Through calculus of variations, the problem of determining the optimally converging basis modes, \( \psi(r) \), can be well-posed in form of an integral equation as shown (Lumley 1967; Delville et al. 1999):

\[
\int_{\Omega} R(r,r')\psi(r')dr' = \lambda \psi(r)
\]  

(B.4)

where, \( R(r,r') \) is the space correlation tensor, defined as:

\[
R(r,r') = \langle q(r,t)q(r',t) \rangle
\]  

(B.5)
In the context of classical application of POD to turbulent structures, the angular brackets denote ensemble average of statistically stationary turbulent flow data. However, the angular brackets in context of the present study interchangeably represent time-averaging of either unsteady RANS or laminar solutions. This approach is often referred in literature as the ‘snapshot POD approach’ (Sirovich 1987; Zhang et al. 2003), where every instantaneous solution is considered as a ‘snapshot’ of the data. Solution of equation (B.4), which is also subjected to a constraint of unitary norm $\| \psi(r) \| = 1$, yields the orthogonal eigenmodes $\psi_n(r)$ with corresponding eigenvalues $\lambda_n$. Consequently, solution of the flow quantity at each time step can be exactly expressed as a linear combination of the eigenmodes (also known as POD modes) in the following form.

$$ q(r,t) = \sum_{n=1}^{N} \phi_n(t) \psi_n(r) $$  \hspace{1cm} (B.6)

The time-dependent multipliers or scalar coefficients, $\phi_n(t)$, are obtained by projecting the solution at each time step on the basis functions as shown.

$$ \phi_n(t) = \int_{\Omega} q(r,t) \psi_n(r) d^{3}r $$  \hspace{1cm} (B.7)

Merely, from a standpoint of equation (B.6), infinite choices for the basis functions $\psi_n(r)$ are available. But, the basis functions determined in conjunction with the minima of $\| F_h(r,t) \|$ ensure optimal and sequential extraction of key features from the data ensemble, enabling truncation of the series (B.6) with fewer POD modes. Thus, succinct expression of a voluminous data ensemble, with focus on its salient aspects is facilitated through the lower-order modes.
B.3 Numerical Implementation

In the current study, the CFD solution at every time step is considered as a snapshot of the flow field, and an ensemble in form of a matrix $A$ is generated as follows:

$$A = [a_1, a_2, a_3, \ldots, a_N]$$  \hspace{1cm} \text{(B.8)}

where, $a_n \in \mathbb{R}^M$; $n = 1, 2, 3, \ldots, N$ denotes the solution at all $M$ nodes in the domain at the $n^{th}$ time step (usually $M > N$). Thus, when $q(r,t)$ is expressed in the above matrix form, the solution of equation (B.4) reduces to obtaining the eigenvalues and eigenvectors of the matrix $AA^T$, which is achieved by the numerical technique of Singular Value Decomposition (SVD).

B.3.1 Singular Value Decomposition (SVD)

Any real matrix $A_{M \times N}$ can be decomposed into the form (Utturkar et al. 2005):

$$A = U\Sigma V^T$$ \hspace{1cm} \text{(B.9)}

where, $U$ is a $(M \times M)$ matrix whose columns form left singular vectors; $V$ is a $(N \times N)$ matrix whose columns form right singular vectors, and $\Sigma$ is a pseudo-diagonal $(M \times N)$ matrix whose diagonal elements are the singular values $s_n$. Furthermore, it can be easily shown that the eigenvalue decomposition of the matrix $AA^T$ can be expressed as:

$$AA^T = U\Sigma^2 U^T$$ \hspace{1cm} \text{(B.10)}

From equations (B.9) and (B.10), it is evident that eigenvalues of $AA^T$ are squares of the singular values of $A$ ($\lambda_n = s_n^2$), and its eigenvectors are the left singular vectors of $A$. Thus, SVD of the ensemble matrix $A$ can effectively yield the desired POD basis functions and their respective eigenvalues.
B.3.2 Post-processing the SVD Output

The SVD subroutine returns the matrix $U$ with $M \times M$ elements. The matrix $U$ is reshaped by extracting only its first $N$ columns. The resultant matrix $U$, which is now an $M \times N$ matrix, comprises the $N$ eigenmodes corresponding to the original data in matrix $A$. Note that the constraint $\| \psi(r) \| = 1$ manifests itself in the following form

$$U^T U = I \quad (B.11)$$

where, $I$ is an $N \times N$ identity matrix. The coefficient matrix for $\phi_n(t)$, in compliance with equation (B.7), is further obtained as:

$$\Gamma = U^T A \quad (B.12)$$

Subsequently, the truncated series with $k$ eigenmodes, in form of another matrix, $\hat{A}(k)$, is constructed as follows:

$$\hat{A}(k) = \hat{U}_{M \times k} \hat{\Gamma}_{k \times N} \quad (B.13)$$

where, $\hat{U} \in U$ and $\hat{\Gamma} \in \Gamma$. It is thus straightforward to observe that the error/discrepancy between the actual and reconstructed data is

$$E(k) = A - \hat{A}(k) \quad (B.14)$$

The fraction of cumulative energy captured by $k$ eigenmodes is calculated as follows (Ahlman et al. 2002; Zhang et al. 2003):

$$E_k = \frac{\sum_{n=1}^{k} s_n^2}{\sum_{n=1}^{N} s_n^2} = \frac{\sum_{n=1}^{k} \lambda_n}{\sum_{n=1}^{N} \lambda_n} \quad (B.15)$$

The energy fractions and the error matrix (equations (B.14) and (B.15)) are commonly used parameters to judge a POD representation.
B.4 Flowchart

B.5 Computing Efforts

In the first step, the algorithm to reduces $A$ to upper bi-diagonal form $U^TAV=B$ by Householder bi-diagonalization. It requires $2MN^2 - (2/3)N^3$ flops. In the second step, diagonalization of the bi-diagonal form requires $14N$ flops.
The SVD subroutine (shown in dotted rectangle) is adopted from Press et al. (1992) which is essentially a Fortran-90 code. Creating subroutines to read the CFD data into matrix $A$, determine the coefficient matrix, reconstruct the truncated series, perform various error estimates and output the solution, constitutes a major part of the present effort. The Fast Fourier Transform (FFT) and least square analyses are performed on MATLAB and MATHCAD.
LIST OF REFERENCES


Lecoffre, Y., 1999, Cavitation Bubble Trackers, A.A. Balkema, Brookfield, VT.


BIOGRAPHICAL SKETCH

Yogen Utturkar was born in Mumbai (formerly Bombay), India, on June 12, 1977. He obtained his B.E. (Bachelor of Engineering) degree in mechanical engineering from the University of Mumbai in May 1999. He worked as a software engineer from 1999-2000 before proceeding to pursue graduate studies in the United States. Yogen obtained his master’s degree in mechanical engineering (minor in engineering sciences) at the University of Florida in April 2002. He investigated synthetic jet flow fields for his M.S. thesis under the tutelage of Dr. Rajat Mittal.

Yogen worked towards his doctoral dissertation under the guidance of Dr. Wei Shyy (chair) and Dr. Nagaraj Arakere (cochair). His research interests comprised numerical modeling of thermodynamic effects in cryogenic cavitation, reduced-order description of simulated data by Proper Orthogonal Decomposition (POD), and development of efficient algorithms for problems involving phase change.

Yogen met his wife, Neeti Pathare, during his graduate years. They married in May 2002 and completed their Ph.D. degree from the same institution (University of Florida).