AEROACOUSTIC COMPUTATION OF TONES GENERATED FROM LOW MACH NUMBER CAVITY FLOWS, USING A PRECONDITIONED METHOD

A Dissertation in Acoustics
by
Brent S. Paul

© 2016 Brent S. Paul

Submitted in Partial Fulfillment of the Requirements for the Degree of Doctor of Philosophy

December 2016
The dissertation of Brent S. Paul was reviewed and approved* by the following:

Philip J. Morris  
Boeing / A.D. Welliver Professor of Aerospace Engineering  
Dissertation Advisor, Chair of Committee

Victor W. Sparrow  
Director of the Graduate Program in Acoustics, Professor of Acoustics

Lyle N. Long  
Distinguished Professor of Aerospace Engineering

Laura L. Pauley  
Professor of Mechanical Engineering

*Signatures are on file in the Graduate School.
Abstract

The hydrodynamically generated noise produced from flow over cavities includes both broadband and tones. The frequency content and amplitude of the resulting noise is a function of the cavity geometry and the approaching boundary layer. The cavity length to depth ratio ($L/D$) is an important parameter that governs the characteristics of cavity noise generated. While both of the noise components are important this work will focus on the production of cavity tones. Cavity tones typically have higher sound pressure levels and can propagate over longer distances than the broadband noise.

The enhancements to the numerical code shown in this work result in the first non-hybrid tool for the prediction of low speed cavity noise. At moderate subsonic Mach numbers the direct calculation of cavity tones has been performed by numerous researchers using highly accurate spatial and time discretization. However, most researchers that are trying to predict the noise from low Mach number flows take a hybrid approach where the fluid dynamics of the simulation are solved with a computational fluid dynamics (CFD) solver and the acoustics are solved separately. The other solver is often based on Lighthill’s Acoustic Analogy or an asymptotic method such as the Expansion about Incompressible Flow (EIF). This work calculates the conservative Navier-Stokes variables to directly predict the cavity tones.

The numerical solver CHOPA (Compressible, High-Order Parallel Acoustics) is extended in this work for the accurate and fast calculation of low Mach number cavity flows. A time-derivative preconditioner equalizes the acoustic wave and turbulence convective speeds to allow for a more efficient time step and shorter calculation times. Because the preconditioner destroys the time accuracy of the solution a dual-time step approach is used for the time integration. Other modifications to the code are required to facilitate the proper implementation of the preconditioner: Matrix-based artificial dissipation, buffer zone, and extrapolation boundary condition. An extension by Buelow of Choi-Merkle’s viscous preconditioner is selected for this work.
There are several different numerical validations performed on the preconditioned Navier-Stokes solver to ensure high quality solutions. First, the combination buffer zone/extrapolation boundary condition is tested by simulating the propagation of a Gaussian pressure pulse. Then the preconditioner is tested with several different analyses. The convection of a uniform velocity flow field with a random perturbation imposed on the flow field tests if the preconditioned solution is independent of the flow Mach number. Then a time accurate Gaussian pressure pulse tests the ability of the preconditioner to solve a time dependent solution. Lastly, a laminar boundary layer flow is calculated and compared to an exact solution showing that the preconditioner is effective for viscous flows.

The prediction of cavity tones from a deep ($L/D = 0.78$) and shallow ($L/D = 2.35$) cavity is simulated for comparison against the experimental measurements of Block. The Mach number of the simulations varied from 0.05 to 0.4. The cavity tone frequencies have an acceptable comparison against the measurements for the deep cavity. However, the shallow cavity tones were almost independent of the flow speed, which may be an indication that standing waves in the cavity could be responsible for the tones for this geometry.

The other cavity simulations replicated the experiment by Stallings et al. for $L/D = 5.42$ and $L/D = 6.25$ for a Mach number of 0.2. The time-averaged wall pressure fluctuations were compared to measurements. While the predicted wall pressures did not match the experiment the discrepancy is because of the existence of a “wake mode” in the numerical results. This is a two-dimensional phenomenon where a large vortex is generated in the cavity and then violently ejected from the cavity, significantly increasing drag. While not matching the experiment the results behave as expected for a cavity resonating in a wake mode.
# Table of Contents

List of Figures x

List of Tables xvii

List of Symbols xviii

Acknowledgments xxix

Chapter 1

Introduction 1

1.1 Theory of Computational Aeroacoustics . . . . . . . . . . . . . . . . 3
1.2 Cavity Noise . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 6
  1.2.1 Cavity Noise Mechanisms . . . . . . . . . . . . . . . . . . . . 6
  1.2.1.1 Shallow Cavities . . . . . . . . . . . . . . . . . . . . 8
  1.2.1.2 Deep Cavities . . . . . . . . . . . . . . . . . . . . . 9
1.2.2 Review of Theoretical Cavity Noise . . . . . . . . . . . . . . . 11
1.2.3 Review of Low Mach Number Cavity Simulations . . . . . . 13
1.2.4 Review of Low Mach Number Cavity Experiments . . . . . . 16
1.3 Objective of Current Study . . . . . . . . . . . . . . . . . . . . . . 18

Chapter 2

Mathematical Formulation 20

2.1 The Governing Equations . . . . . . . . . . . . . . . . . . . . . . . 20
2.2 Numerical Discretization . . . . . . . . . . . . . . . . . . . . . . . 25
  2.2.1 Spatial Discretization . . . . . . . . . . . . . . . . . . . . . . 25
  2.2.2 Time Discretization . . . . . . . . . . . . . . . . . . . . . . . 26
    2.2.2.1 Dual Time-Step . . . . . . . . . . . . . . . . . . . . 26
    2.2.2.2 Explicit Time Integration . . . . . . . . . . . . . . 28
    2.2.2.3 Implicit Time Integration . . . . . . . . . . . . . . 29
  2.2.4 Calculation of Time Step . . . . . . . . . . . . . . . . . . . . 30
2.3 Convergence Acceleration ........................................ 34
2.4 Preconditioning .................................................... 34
  2.4.1 Time Derivative Preconditioner ............................... 36
    2.4.1.1 Incompressible Preconditioner ......................... 37
    2.4.1.2 Compressible Preconditioner ......................... 39
  2.4.2 Asymptotic Expansion Preconditioner ...................... 41
  2.4.3 Inviscid Preconditioner .................................... 43
    2.4.3.1 Turkel Preconditioner ................................ 43
    2.4.3.2 van Leer Preconditioner ............................... 44
    2.4.3.3 Choi-Merkle Preconditioner ......................... 46
  2.4.4 Navier-Stokes Preconditioner .............................. 47
    2.4.4.1 Time Derivative Preconditioner ....................... 49
      2.4.4.1.1 Merkle Preconditioner ............................ 49
      2.4.4.1.2 Weiss/Smith Preconditioner ...................... 50
      2.4.4.1.3 Pletcher/Chen Preconditioner .................... 51
    2.4.4.2 Asymptotic Preconditioner Section .................... 52
      2.4.4.2.1 Merkle 1988 ...................................... 52
      2.4.4.2.2 Merkle Asymptotic ............................... 53
  2.4.4.3 Other Viscous Preconditioners ......................... 55
    2.4.4.3.1 Viscous Jacobi Preconditioner ..................... 56
  2.4.4.4 Preconditioner of Choice ............................... 58
  2.4.4.5 Pressure/Density Decomposition ......................... 60
  2.4.4.6 Selection of Preconditioning Parameter, \( \varepsilon \) ... 62
  2.4.5 Boundary Conditions ....................................... 64
    2.4.5.1 Method of Characteristics ............................ 65
    2.4.5.2 Extrapolation Boundary Condition .................... 70
  2.4.5.3 Buffer Zone ............................................. 71
    2.4.5.3.1 WGK Buffer Zone ................................ 71
    2.4.5.3.2 JBF Buffer Zone ................................ 73
  2.4.6 Artificial Dissipation ..................................... 75
  2.5 Turbulence Modeling .......................................... 82
    2.5.1 Spalart-Allmaras One-Equation Model .................... 83
    2.5.2 Detached-Eddy Simulation ............................... 85

Chapter 3
Code Validation .................................................. 88
  3.1 Boundary Conditions .......................................... 88
    3.1.1 Existing Inflow/Outflow Boundary Conditions ............ 88
      3.1.1.1 Riemann Invariant Boundary Condition ............. 88
Chapter 4
Two-Dimensional Cavity Simulations

4.1 Block Experiment
  4.1.1 Experimental Parameters and Measurements
  4.1.2 Numerical Grids
    4.1.2.1 L/D=0.78 Grid
    4.1.2.2 L/D=2.35 Grid
  4.1.3 Solution Parameters
  4.1.4 FFT Calculation Methodology
  4.1.5 L/D = 0.78 Cavity Solution
    4.1.5.1 Pressure Spectra
    4.1.5.2 Modifications to Artificial Dissipation and Grid Density For Improved Results
    4.1.5.3 Modifications to Convergence Criteria Improved Results
    4.1.5.4 Convection Velocity
    4.1.5.5 Comparison to Equations
    4.1.5.6 Cavity Tone Conclusion
    4.1.5.7 Flow Results
    4.1.5.8 Timing
    4.1.5.9 L/D = 0.78 Conclusions
  4.1.6 L/D = 2.35 Cavity Solution
    4.1.6.1 Pressure Spectra
    4.1.6.2 Convection Velocity
    4.1.6.3 Cavity Tone Conclusion
    4.1.6.4 Flow Results
    4.1.6.5 Timing
    4.1.6.6 L/D = 2.35 Conclusions

4.2 Stallings Experiment
  4.2.1 Experimental Parameters and Measurements
  4.2.2 Numerical Grids
    4.2.2.1 L/D=5.42 Grid
    4.2.2.2 L/D=6.25 Grid
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.2.3 Solution Parameters</td>
<td>182</td>
</tr>
<tr>
<td>4.2.4 L/D=6.25 Cavity Solution</td>
<td>184</td>
</tr>
<tr>
<td>4.2.4.1 Flow Results</td>
<td>184</td>
</tr>
<tr>
<td>4.2.4.2 Time-Averaged Wall Pressure Fluctuations</td>
<td>188</td>
</tr>
<tr>
<td>4.2.5 L/D=5.42 Cavity Solution</td>
<td>189</td>
</tr>
<tr>
<td>4.2.5.1 Flow Results</td>
<td>190</td>
</tr>
<tr>
<td>4.2.5.2 Time-Averaged Wall Pressure Fluctuations</td>
<td>191</td>
</tr>
<tr>
<td>4.2.6 Wake Mode</td>
<td>191</td>
</tr>
<tr>
<td>4.2.7 Timing</td>
<td>199</td>
</tr>
<tr>
<td>4.2.8 Conclusions</td>
<td>200</td>
</tr>
<tr>
<td>4.3 Conclusions</td>
<td>201</td>
</tr>
<tr>
<td>Chapter 5</td>
<td></td>
</tr>
<tr>
<td>Summary, Conclusions and Directions for Future Research</td>
<td>202</td>
</tr>
<tr>
<td>5.1 Summary and Conclusions</td>
<td>202</td>
</tr>
<tr>
<td>5.2 Suggestions For Further Studies</td>
<td>207</td>
</tr>
<tr>
<td>Appendix A</td>
<td></td>
</tr>
<tr>
<td>Governing Equations in Computational Space</td>
<td>209</td>
</tr>
<tr>
<td>A.1 Three Dimensions</td>
<td>209</td>
</tr>
<tr>
<td>A.2 Two Dimensions</td>
<td>213</td>
</tr>
<tr>
<td>Appendix B</td>
<td></td>
</tr>
<tr>
<td>Dispersion Relation Preserving Coefficients</td>
<td>215</td>
</tr>
<tr>
<td>Appendix C</td>
<td></td>
</tr>
<tr>
<td>Preconditioner Information</td>
<td>216</td>
</tr>
<tr>
<td>Appendix D</td>
<td></td>
</tr>
<tr>
<td>Preconditioner Matrices</td>
<td>218</td>
</tr>
<tr>
<td>D.1 Incompressible Preconditioners</td>
<td>218</td>
</tr>
<tr>
<td>D.2 Euler Preconditioners</td>
<td>219</td>
</tr>
<tr>
<td>D.2.1 Turkel Preconditioners</td>
<td>219</td>
</tr>
<tr>
<td>D.2.2 van Leer Preconditioners</td>
<td>223</td>
</tr>
<tr>
<td>D.2.3 Choi Merkle Inviscid</td>
<td>225</td>
</tr>
<tr>
<td>D.2.4 D. Lee Preconditioner</td>
<td>225</td>
</tr>
<tr>
<td>D.2.5 Jacobian Preconditioner</td>
<td>227</td>
</tr>
<tr>
<td>D.3 Navier-Stokes Preconditioners</td>
<td>227</td>
</tr>
<tr>
<td>D.3.1 Viscous Jacobi Preconditioner</td>
<td>227</td>
</tr>
<tr>
<td>D.3.1.1 Turkel Preconditioner</td>
<td>227</td>
</tr>
</tbody>
</table>
D.3.1.2  van Leer Preconditioner .................................. 227
D.3.1.3  Pierce/Giles Preconditioner .............................. 229
D.3.2  Dispersion Analysis Preconditioner ......................... 229
D.3.2.1  Lee Preconditioner ...................................... 229
D.3.2.2  Venkateswaran Merkle Preconditioner ................... 230
D.3.2.3  van Leer Preconditioner .................................. 230
D.3.3  Time Derivative Viscous Preconditioners .................... 231
D.3.3.1  Merkle Preconditioner ................................... 231
D.3.3.2  Weiss/Smith Preconditioner .............................. 232
D.3.4  Link Between CMi / CMv / WS Preconditioners .......... 233
D.3.5  Pletcher/Chen Preconditioner .............................. 233
D.3.6  Asymptotic Preconditioners ................................. 234
D.3.6.1  Merkle 1988 .............................................. 234
D.3.6.2  Merkle Asymptotic ....................................... 234
D.3.7  Hybrid Preconditioner ...................................... 236

Bibliography ......................................................... 237
List of Figures

1.1 (a) Instantaneous density contours and (b) sound pressure levels for a cavity with a length to depth ratio of 2.0 and freestream Mach number of 0.4 [1]................. 5
1.2 Various dimensional parameters for cavity flows............................... 7
1.3 Pseudo-piston model for cavity tones. ..................................... 9
1.4 Physical mechanism for a cavity tone for a deep cavity................. 10

2.1 Interdependency of physical time (t) and pseudo-time (τ) in a dual time-stepping scheme. ................................................................. 27
2.2 Buffer zone applied to all boundaries. ........................................ 72
2.3 Plot of JBF buffer zone along y = 0. F1 is the magnitude of the damping function............................................................. 72

3.1 Grid used to validate buffer zone and extrapolation boundary condition. Every other grid point has been omitted for clarity .................... 90
3.2 Propagation of two dimensional acoustic pulse. The colored lines are: red is β = 2, green is β = 3, blue is β = 4, and black is the exact solution. .................................................. 91
3.3 Initial streamwise velocity distribution for Mach number of 0.25. .... 92
3.4 Residual for the three CHOPA configurations for Mach number of 0.5. ................................................................................. 94
3.5 Residual for the three CHOPA configurations for Mach number of 0.1. ................................................................................. 95
3.6 Residual for the three CHOPA configurations for Mach number of 0.01. The unpreconditioned version converges at 38360 iterations. . 96
3.7 Residual for the three CHOPA configurations for Mach number of 0.001. ............................................................................. 97
3.8 Residual for the three CHOPA configurations for Mach number of 0.0001. ................................................................. 97
3.9 Grid used for the simulation of the propagation of a Gaussian pulse.  
Every other grid point is omitted for clarity.  

3.10 Initial condition for density along centerline.  

3.11 Nondimensional density for the exact, unpreconditioned, and preconditioned solution at four different time steps with a freestream Mach number of 0.0.  

3.12 Nondimensional density for the exact, unpreconditioned, and preconditioned solution at four different time steps with a freestream Mach number of 0.01.  

3.13 Computational grid and boundary conditions for the flat plate simulation. The grey area represents the buffer zone region and the red area is the undamped simulation area.  

3.14 Nondimensional density for the exact, unpreconditioned, and preconditioned solution at $x/L = 1$ with a freestream Mach number of 0.4.  

3.15 Nondimensional axial velocity for the exact, unpreconditioned, and preconditioned solution at $x/L = 1$ with a freestream Mach number of 0.4.  

3.16 Nondimensional density for the exact, unpreconditioned, and preconditioned solution at $x/L = 1$ with a freestream Mach number of 0.1.  

3.17 Nondimensional axial velocity for the exact, unpreconditioned, and preconditioned solution at $x/L = 1$ with a freestream Mach number of 0.1.  

3.18 Nondimensional density for the exact, unpreconditioned, and preconditioned solution at $x/L = 1$ with a freestream Mach number of 0.01.  

3.19 Nondimensional axial velocity for the exact, unpreconditioned, and preconditioned solution at $x/L = 1$ with a freestream Mach number of 0.01.  

3.20 Nondimensional density for the unpreconditioned solution with and without a buffer zone. The solution is plotted at $x/L = 1$ and the freestream Mach number is 0.1.  

3.21 Nondimensional velocity for the unpreconditioned solution with and without a buffer zone. The solution is plotted at $x/L = 1$ and the freestream Mach number is 0.1.  

4.1 Cavity tone frequencies for varying Mach numbers. This is Figure 4 from Reference [2].  

4.2 Grid used for $L/D = 0.78$ cavity.
4.3 Locations of pressure measurements for $L/D = 0.78$ cavity.

4.4 Grid used for $L/D = 2.35$ cavity.

4.5 Locations of pressure measurements for $L/D = 2.35$ cavity.

4.6 Buffer zone for $L/D = 0.78$ cavity.

4.7 Buffer zone for $L/D = 2.35$ cavity.

4.8 Velocity profiles for $L/D = 0.78$ at Mach number 0.1.

4.9 Velocity profiles for $L/D = 0.78$ at Mach number 0.3.

4.10 Velocity profiles for $L/D = 2.35$ at Mach number 0.1.

4.11 Velocity profiles for $L/D = 2.35$ at Mach number 0.3.

4.12 Spectra with varying FFT parameters.

4.13 Spectra with varying FFT parameters.

4.14 Spectra with varying FFT parameters.

4.15 Nondimensional spectral density for $M = 0.05$. (a) All pressure history locations (b) Five select pressure history locations.

4.16 Nondimensional spectral density for $M = 0.1$. (a) All pressure history locations (b) Five select pressure history locations.

4.17 Nondimensional spectral density for $M = 0.15$. (a) All pressure history locations (b) Five select pressure history locations.

4.18 Nondimensional spectral density for $M = 0.2$. (a) All pressure history locations (b) Five select pressure history locations.

4.19 Nondimensional spectral density for $M = 0.3$. (a) All pressure history locations (b) Five select pressure history locations.

4.20 Nondimensional spectral density for $M = 0.4$. (a) All pressure history locations (b) Five select pressure history locations.

4.21 Nondimensional spectral density for $M = 0.4$. (a) All pressure history locations (b) Five select pressure history locations.

4.22 Nondimensional spectral density for $M = 0.05$ for lower artificial dissipation coefficient. (a) All pressure history locations (b) Five select pressure history locations.

4.23 Nondimensional spectral density for $M = 0.05$ with an increased vertical distribution of grid points. (a) All pressure history locations (b) Five select pressure history locations.

4.24 Nondimensional spectral density for $M = 0.05$ for all points (a) Original calculation parameters (b) Increased cavity grid density and reduced artificial dissipation coefficient.

4.25 Nondimensional spectral density for $M = 0.1$ for all points (a) Original calculation parameters (b) Increased cavity grid density and reduced artificial dissipation coefficient.
4.26 Nondimensional spectral density for \( M = 0.3 \) for selected points (a)
Maximum of 500 subiterations (b) Maximum of 5000 subiterations
(standard) .......................................................... 138
4.27 Comparison of residual for different ERRMINs. .................. 139
4.28 Nondimensional spectral density of pressure for \( M = 0.3 \) for selected
points. The solution is using an unpreconditioned method. (a)
Maximum of 500 subiterations (b) Maximum of 5000 subiterations
(standard) .......................................................... 140
4.29 Normalized cross correlation coefficients for pressure between cavity
leading and trailing edges for \( M = 0.4 \) using an unpreconditioned
method. .......................................................... 141
4.30 Spatial separation relative to time delay for \( M = 0.4 \) using an un-
preconditioned method. ............................................ 143
4.31 Data for convection velocity of \( M = 0.05 \) solution. (a) Normalized
cross correlation coefficients. (b) Spatial separtion relative to time
delay .......................................................... 144
4.32 Data for convection velocity of \( M = 0.1 \) solution. (a) Normalized
cross correlation coefficients. (b) Spatial separtion relative to time
delay .......................................................... 144
4.33 Data for convection velocity of \( M = 0.15 \) solution. (a) Normalized
cross correlation coefficients. (b) Spatial separtion relative to time
delay .......................................................... 145
4.34 Data for convection velocity of \( M = 0.2 \) solution. (a) Normalized
cross correlation coefficients. (b) Spatial separtion relative to time
delay .......................................................... 145
4.35 Data for convection velocity of \( M = 0.3 \) solution. (a) Normalized
cross correlation coefficients. (b) Spatial separtion relative to time
delay .......................................................... 146
4.36 Data for convection velocity of \( M = 0.3 \) unpreconditioned solution.
(a) Normalized cross correlation coefficients. (b) Spatial separtion
relative to time delay ............................................ 147
4.37 Data for convection velocity of \( M = 0.4 \) solution. (a) Normalized
cross correlation coefficients. (b) Spatial separtion relative to time
delay .......................................................... 147
4.38 Data for convection velocity of \( M = 0.4 \) unpreconditioned solution.
(a) Normalized cross correlation coefficients. (b) Spatial separtion
relative to time delay ............................................ 148
4.39 Convection velocity at each flow speed. The convection velocity for
\( M = 0.4 \) is for unpreconditioned. .......................... 148
4.40 Comparison of cavity tones between the CHOP A simulation and the prediction from Block’s equations using a convection velocity of $0.57U_\infty$. .................................................. 150
4.41 Comparison of cavity tones between the CHOP A simulation using a higher density grid with a lower artificial dissipation coefficient and the prediction from Block’s equations using a convection velocity of $0.57U_\infty$. .................................................. 151
4.42 Comparison of cavity tones between the CHOP A simulation and the prediction from Block’s equations using the convection velocity from the CHOPA simulation. ........................................ 152
4.43 Comparison of cavity tones between the CHOP A simulation using a higher density grid with a lower artificial dissipation coefficient and the prediction from Block’s equations using the convection velocity from the CHOPA simulation. ........................................ 153
4.44 Nondimensional density at $t = 500$. ........................................ 154
4.45 Nondimensional pressure at $t = 500$. ........................................ 155
4.46 Nondimensional density at $t = 500$. ........................................ 156
4.47 Nondimensional density at $t = 500$. ........................................ 156
4.48 Nondimensional pressure at $t = 500$. ........................................ 157
4.49 Simulation timings for $M = 0.05$ flow speed. ............................. 159
4.50 Simulation timings for $M = 0.3$ flow speed. ............................. 159
4.51 Simulation time for each flow speed. ....................................... 160
4.52 Nondimensional spectral density for $M = 0.05$. (a) All pressure history locations (b) Five select pressure history locations. ............... 163
4.53 Nondimensional spectral density for $M = 0.1$. (a) All pressure history locations (b) Five select pressure history locations. ............... 164
4.54 Nondimensional spectral density for $M = 0.15$. (a) All pressure history locations (b) Five select pressure history locations. ............... 165
4.55 Nondimensional spectral density for $M = 0.2$. (a) All pressure history locations (b) Five select pressure history locations. ............... 165
4.56 Nondimensional spectral density for $M = 0.3$. (a) All pressure history locations (b) Five select pressure history locations. ............... 166
4.57 Nondimensional spectral density for $M = 0.3$ for unpreconditioned solution. (a) All pressure history locations (b) Five select pressure history locations. ............... 167
4.58 Nondimensional spectral density for $M = 0.4$. (a) All pressure history locations (b) Five select pressure history locations. ............... 167
4.59 Spatial separation relative to time delay for $M = 0.05$ solution. .... 169
4.60 Spatial separation relative to time delay for $M = 0.10$ solution. .... 169
4.61 Spatial separation relative to time delay for $M = 0.15$ solution. .... 170
4.80 Time-averaged wall pressure fluctuations measurements for $L/D = 6.25$ cavity. ................................................................. 189
4.81 Wall pressures on cavity floor for different length cavities. This is Figure 7 from Reference [3]. ...................................................... 190
4.82 $L/D = 5.42$ contours of nondimensional pressure at four different time steps using a preconditioned method. ......................... 191
4.83 $L/D = 5.42$ contours of nondimensional pressure at four different time steps using an unpreconditioned method. ......................... 192
4.84 $L/D = 5.42$ contours of Mach number at four different time steps using a preconditioned method. .............................................. 193
4.85 $L/D = 5.42$ contours of Mach number at four different time steps using an unpreconditioned method. .............................................. 194
4.86 Time-averaged wall pressure fluctuations measurements for $L/D = 5.42$ cavity for $M = 0.2$. ...................................................... 195
4.87 Reprint of Gharib and Roshko’s Figure 17 showing difference in pressure coefficient for two different $L/\theta$ conditions [4]. .................. 196
4.88 Comparison of the time-averaged streamlines of $\rho u$ for Shieh and Morris (top left) and CHOPA (bottom right). .............................. 197
4.89 Time-averaged contours of $\rho u$ for $L/D = 5.42$ with Mach number $= 0.2$ flow. ................................................................. 197
4.90 Time-averaged contours of $\rho u$ for $L/D = 6.25$ with Mach number $= 0.2$ flow. ................................................................. 198
4.91 Time-averaged contours of $\rho u$ for $L/D = 5.42$ with Mach number $= 0.1$ flow. ................................................................. 198
4.92 Total calculation times for the Stallings cavity simulations. The $M = 0.1$ solution is only for 1000 time steps. ................................. 199
## List of Tables

1.1 Approximate regimes for shallow cavity based on geometry. 8  
1.2 Experimental database for low Mach number cavity flows. 17  

3.1 Description of test cases to validate inviscid preconditioner. \( d_6 \) is the coefficient of artificial dissipation. 93  
3.2 Convergence histories for uniform flow problem. Cases 1 through 5 are unpreconditioned. *The unpreconditioned problem for Mach number of 0.001 was unable to converge. 99  
3.3 Preconditioner effectiveness. N/A signifies a non-physical solution for the unpreconditioned code. A negative reduction means the preconditioned code took longer to run. 101  

4.1 Buffer zone parameters for \( L/D = 0.78 \) cavity. 123  
4.2 Buffer zone parameters for \( L/D = 2.35 \) cavity. 123  
4.3 Nondimensional boundary layer thickness \((x/L_{0.78})\). 125  
4.4 Ratio of computational tone frequencies to the experimental values 134  
4.5 Buffer zone parameters for \( L/D = 0.78 \) cavity. 140  
4.6 Nondimensional boundary layer thickness. 172  
4.7 \( L/D = 2.35 \) convection velocities and cavity tone Strouhal numbers for all flow speeds. 179  
4.8 Buffer zone parameters for \( L/D = 5.42 \) and \( L/D = 6.25 \) cavities. 183
## List of Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>A, B, C</td>
<td>inviscid flux Jacobian matrix</td>
</tr>
<tr>
<td>$A_v$, $B_v$, $C_v$</td>
<td>viscous flux Jacobian matrix</td>
</tr>
<tr>
<td>$\mathcal{A}$</td>
<td>artificial dissipation</td>
</tr>
<tr>
<td>$c$</td>
<td>speed of sound</td>
</tr>
<tr>
<td>$c'$</td>
<td>preconditioned speed of sound</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant-Friedrich-Lewy number, $\lambda \Delta t/\Delta x$</td>
</tr>
<tr>
<td>$E, F, G$</td>
<td>inviscid or viscous flux matrices</td>
</tr>
<tr>
<td>$E_v, F_v, G_v$</td>
<td>viscous flux matrices</td>
</tr>
<tr>
<td>$e$</td>
<td>internal energy per unit volume</td>
</tr>
<tr>
<td>$E_t$</td>
<td>total energy per unit volume</td>
</tr>
<tr>
<td>$h$</td>
<td>enthalpy</td>
</tr>
<tr>
<td>$H$</td>
<td>Navier-Stokes source vector</td>
</tr>
<tr>
<td>$h_0$</td>
<td>stagnation enthalpy or total enthalpy per unit volume</td>
</tr>
<tr>
<td>$i$</td>
<td>$\sqrt{-1}$</td>
</tr>
<tr>
<td>$J$</td>
<td>Jacobian</td>
</tr>
<tr>
<td>$k$</td>
<td>wavenumber</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number</td>
</tr>
<tr>
<td>$p, p_0$</td>
<td>thermodynamic pressure</td>
</tr>
<tr>
<td>$\Pr$</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>$q$</td>
<td>Total velocity $q = \sqrt{(u^2 + v^2 + w^2)}$</td>
</tr>
<tr>
<td>$q_i$</td>
<td>heat flux in i direction</td>
</tr>
<tr>
<td>$Q$</td>
<td>solution vector, $(\rho, \rho u, \rho v, \rho w, \rho E_t)^T$</td>
</tr>
<tr>
<td>$Q_v$</td>
<td>solution vector, $(\rho, u, v, w, T)^T$</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$Re_{\Delta x}$</td>
<td>cell Reynolds number</td>
</tr>
<tr>
<td>$Re_c$</td>
<td>acoustic Reynolds number</td>
</tr>
<tr>
<td>$S$</td>
<td>entropy</td>
</tr>
<tr>
<td>$S_L, S_R$</td>
<td>left and right eigenvectors</td>
</tr>
</tbody>
</table>

xviii
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sr</td>
<td>Strouhal number</td>
</tr>
<tr>
<td>t</td>
<td>time</td>
</tr>
<tr>
<td>T</td>
<td>temperature</td>
</tr>
<tr>
<td>U</td>
<td>mean flow or contravariant velocity</td>
</tr>
<tr>
<td>u, v, w</td>
<td>convective speed in x-, y-, and z-direction</td>
</tr>
<tr>
<td>VNN</td>
<td>von Neumann number, $\nu \Delta t/(\Delta x)^2$</td>
</tr>
<tr>
<td>x, y, z</td>
<td>Cartesian coordinate directions</td>
</tr>
</tbody>
</table>

**Greek:**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha, \beta, \delta, \tau, \psi$</td>
<td>preconditioning parameters</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>artificial dissipation scaling factor</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>preconditioning limiter, perturbation expansion parameter</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>ratio of specific heats</td>
</tr>
<tr>
<td>$\Gamma$</td>
<td>preconditioning matrix</td>
</tr>
<tr>
<td>$\lambda_i$</td>
<td>eigenvalue of $i^{th}$ direction</td>
</tr>
<tr>
<td>$\mu$</td>
<td>molecular viscosity</td>
</tr>
<tr>
<td>$\mu_B$</td>
<td>bulk viscosity</td>
</tr>
<tr>
<td>$\nu$</td>
<td>kinematic viscosity</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>spectral radius, preconditioning parameter,</td>
</tr>
<tr>
<td>$\beta$</td>
<td>JBF buffer zone damping function</td>
</tr>
<tr>
<td>$\varsigma$</td>
<td>WGK buffer zone damping function</td>
</tr>
<tr>
<td>$\tau_{ij}$</td>
<td>stress tensor</td>
</tr>
<tr>
<td>$\nabla$</td>
<td>backward difference operator</td>
</tr>
<tr>
<td>$\Delta$</td>
<td>forward difference operator</td>
</tr>
<tr>
<td>$\xi, \eta, \zeta$</td>
<td>generalized coordinate directions</td>
</tr>
</tbody>
</table>

**Subscripts:**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$i$</td>
<td>tensor notation</td>
</tr>
<tr>
<td>$r$</td>
<td>reference value</td>
</tr>
<tr>
<td>$x, y$</td>
<td>coordinate directions</td>
</tr>
<tr>
<td>$</td>
<td></td>
</tr>
<tr>
<td>$\infty$</td>
<td>freestream value</td>
</tr>
<tr>
<td>$\prime$</td>
<td>perturbation value</td>
</tr>
</tbody>
</table>
Overbar:
- artificial variable
I would like to thank my committee members for their guidance and encouragement throughout my research: Dr. Philip Morris, Dr. Lyle Long, Dr. Laura Pauley, and Dr. Victor Sparrow. I am lucky to have had the chance to learn from them and this thesis is much better as a result. I would also like to thank Dr. Vigor Yang who served on my committee before leaving Penn State. Beyond his role as a committee member I would like to specifically thank my advisor, Dr. Morris, for his unwavering support and guidance. During the many years I have spent working on both of my graduate degrees he has given me direction and advice for many years and I am truly grateful.

As a student who worked full time during this thesis there are also numerous colleagues I would like to thank as well. First, and foremost, is Gary Cooper at Electric Boat for providing the impetus for me to embark on this journey. Without his support I would not have been able to take advantage of a great opportunity and this research would not have happened. I would also like to thank Chris Hoddinott for his encouragement as well as Joe Gavin and Scot Slimon for their expertise and advice. There are many others in the EB Signatures Department that I have had numerous conversations with over the years that have helped to improve and shape this thesis. Conversations with Dr. Ted Farabee at the Naval Surface Warfare Center, Carderock Division were useful in shaping the initial direction of this research. Additional thanks goes to my colleagues at Alion Science who have pushed me to complete my research. In addition to other students at Penn State, Dr. Chingwei Shieh was a great friend who shared his knowledge of the CHOPA code and computational aeroacoustics. Since our undergraduate classes our conversations have been formative and his encouragement was invaluable.

Lastly, I would not have completed this thesis without the support of my family and friends. Thank you to my parents for instilling in me a desire to further my education. Thank you to my friends who have encouraged me to finish my degree. And thank you most of all to my wife, Roberta, for encouraging me to complete what has been a long and arduous task. Lastly, I would like to dedicate this to
my children, Brodie and Jenna, to show them they can reach their goals through dedication and hard work.

This research was partially supported by the Electric Boat Corporation. Other support was also given by the Office of Naval Research (ONR) under the program management of Dr. Luise Couchman.
Flow past cavities can be found on a wide range of vehicles, covering both aerospace and marine platforms [5–7]. These cavities can present both a fluid dynamic and acoustical concern. For some applications the flow over the cavity can be severe enough to cause buffeting of the space within the cavity. Additionally, the flow over the cavity can generate both broadband noise and tones. Due to the continually decreasing noise requirements for aircraft, open cavities have received significant attention, both numerically and experimentally in the past few decades. The most commonly examined application has been for aircraft where bomb bays as well as wheel wells are geometries of concern. For open bomb bays the flow over the cavity can be such that it can prevent the separation of the weapon stores. For commercial aircraft, during landing, the open wheel well can generate significant broadband noise that is perceived by the community on the ground. For most cavity geometries significant tones occur in addition to the broadband noise.

Hydrodynamically generated noise from flow over cavities at lower Mach numbers is also important. The sunroof has transitioned from a luxury to an essential item on automobiles, but can be a source of pressure fluctuations that cause discomfort to passengers. As such, there has been an increase in mitigation techniques for sunroofs [8, 9]. Unfortunately, at the lower speeds associated with automobiles there are difficulties with numerical simulation, so experiments are usually required.

For vehicles in water (e.g. submarines, underwater unmanned vehicles, and surface ships) the Mach numbers are even lower, equivalent to 0.01, and the density of water makes it difficult to simulate the acoustics of this almost incompressible flow [10, 11]. The functions dictated for naval vessels require the ability to perform
a wide range of missions, while retaining the ability to remain quiet. During the
design of the vessel, as well as when making improvements during the vessel’s life
cycle, the resulting flexibility must be traded off with any potential degradation
in the acoustic performance of the platform. For sound that is produced by hyd-
dynamic forces, the geometry and location of a cavity in the flow field are mostly
responsible for determining the radiated noise spectra.

Currently, predictions for hydrodynamically induced sound are performed at
a high level using scaling laws that can be used in a wide variety of applications.
The physical mechanisms of flow induced sound are understood for many condi-
tions, but the complex flow field often permits only a qualitative prediction using
scaling laws. These predictions usually rely on empirical parent curves, which can
unnecessarily restrict the prediction due to the unique characteristics of the par-
tent curve. This can result in uncertainty in the acoustic analysis of the design.
A method to quantitatively assess a design is to reproduce the flow field with a
small scale model and measure the radiated noise. This is often cumbersome and
expensive, with the costs often outweighing the benefits of the test. Improved
methods must be developed that will allow quick and efficient acoustic input to
design decisions. A computational model that can predict the generation of noise
from the hydrodynamic flow field possesses this capability and will improve the
design process.

To assist in the design of vehicles moving at a low Mach number the focus of the
present study is on the prediction of directly radiated cavity tones hydrodynamically
generated by flow. The capability to predict the broadband noise from flow
over cavities exists, but it requires detailed examination of the turbulence models
used in the simulation. As such, the prediction of the frequency and amplitude of
cavity tones will be presented in this work.

The modifications made to the CHOPA code and the results shown in this
work represent that first non-hybrid tool that is for the prediction of low speed
cavity noise. Previous researchers have calculated cavity noise using a high-order
computational aeroacoustics code but those have been for moderate subsonic Mach
numbers [12–14].

Computational aeroacoustic approaches for low Mach number flows typically
use a hybrid approach where the acoustic solution is solved separately from the
fluid solution. There are two hybrid methods that are used most often. One hybrid
approach solves for the acoustic source strengths and distributions using a fluid
dynamic solver and then an acoustic wave equation (typically Lighthill’s Acoustic
Analogy) to propagate the noise to the far field [15, 16]. The other approach is
an asymptotic method where the acoustic source terms are calculated from the
incompressible flow equations called the Expansion about Incompressible Flow
(EIF). The velocity and pressure components are segregated into incompressible
and perturbation terms. The small vortical length scales are then calculated from
an incompressible flow solver and the large scale acoustic modes are predicted from
the perturbation equations [17].

1.1 Theory of Computational Aeroacoustics

It is often prohibitively expensive to test different cavity designs experimentally,
regardless of the fluid medium (air or water). As such, and with increases in
computational power, numerical simulations of cavity flows are now possible for
engineering analyses. Since the noise produced by cavity flows is of interest, here
instead of computational fluid dynamics (CFD), a computational aeroacoustics
(CAA) approach is warranted [18]. This is different from CFD solutions in several
ways:

• The acoustic pressure is often several of orders of magnitude lower than
  the flow perturbations and requires high-order spatial discretizations for its
  simulation.

• Time dependency. By their nature, pressure waves are unsteady and require
  a time accurate solution. Many fluid dynamic problems can be solved with
  a steady-state solver, greatly simplifying the solution process.

• Far field boundary conditions. The boundaries of the domain must allow
  both fluid flow and acoustical waves to propagate out of the domain, which
  requires a more robust boundary condition than those requiring just fluid
  flow to leave the domain. Thus, CAA boundary conditions must be more
  rigorous to allow for simulations without having spurious reflections.

• Length scales. The length scales of interest for a CAA problem are those
  associated with acoustic waves while the length scales of interest that char-
acterize the flow field are on the order of the shear layer thickness. The length scales of the source region (i.e., cavity) and the acoustic far field are usually several orders of magnitude different. The CAA solver must be able to handle these different length scales.

CAA has been selected over three other methods: DNS (Direct Numerical Simulation), hybrid, and EIF (Expansion about Incompressible Flow). DNS is a method that solves turbulent flows by directly resolving all scales of the flow, requiring an inordinate number of grid points. The resolution of the method has a grid dependence on Reynolds number, prohibiting it as a solution method for the high Reynolds numbers found for typical cavity flows. The hybrid method combines a highly accurate CFD solver for the hydrodynamic flow with another solver based on an acoustic analogy for the noise. The third method, EIF, was not chosen because it does not directly solve for both the acoustics and fluid dynamics of the flow, but separates the two phenomena using separate equation sets to develop a solution. This splitting of the flow and acoustic field assumes that the propagation of acoustic waves is not affected by viscosity. Also, while the acoustic solution is solved on a separate grid and far field solutions are easier for the EIF approach this advantage can be offset in a CAA method through the use of far field propagation methods.

The CHOPA (Compressible, High-Order, Parallel Acoustics) code is selected as the foundation for development since it solves the Navier-Stokes equations, which encompass both the fluid dynamics and acoustics of the flow field simultaneously. This code was developed by the Pennsylvania State University Computational Aero- and Thermal Acoustics group and has already been validated for the simulation of moderate to high subsonic cavity flows [1, 19–21]. CHOPA solves the compressible unsteady Favre Averaged Navier-Stokes (FANS) equations in either two or three dimensions and was initially designed to predict near field pressure fluctuations for cavity flows. Favre averaging follows the principles of Reynolds averaging but is a density-weighted average and simplifies the compressible equations of motion.

A significant difficulty with the solution of low Mach number cavity flow is due to the large difference between the acoustic and convective wave speeds. As the numerical time step used to integrate the equations in time is proportional to the inverse of the acoustic wave speed, the time step required for stability is very
small. This results in a large number of time steps to convect particles moving at the flow speed, resulting in very long computation times. If the Mach number is low enough this can eventually lead to numerical instability. To combat this problem preconditioning of the governing equations is performed to equalize the wave speeds and allow for an efficient solution.

As seen from Figure 1.1 the flow field within and surrounding cavities is unsteady with significant pressure and velocity fluctuations [1]. In realistic simulations the cavity flow may have important three-dimensional aspects that will need to be modeled for accurate results. As such, typical CFD algorithms are usually of too low an order in both space and time to accurately resolve the fluctuating flow field and acoustics in a cavity simulation. In this thesis a fourth order dispersion-relation-preserving (DRP) scheme [22] is used for the spatial derivatives, while the time integration is performed using a dual-time stepping scheme. This method uses an explicit, fourth-order modified Runge-Kutta method for the inner loop and an implicit, second-order, backward difference for the outer loop. The higher order methods allow for the fluid dynamics and acoustics of the cavity to be resolved accurately, without the dissipation and dispersive problems that occur with low order methods.

Figure 1.1. (a) Instantaneous density contours and (b) sound pressure levels for a cavity with a length to depth ratio of 2.0 and freestream Mach number of 0.4 [1].
1.2 Cavity Noise

Most of the research in cavity noise has been performed in the moderate subsonic to low supersonic flow regime \((M > 0.4)\), with the impetus being the aircraft industry; both military (bomb bays for weapon storage) and civilian (acoustic regulations for landing and take-off). The following sections describe the mechanisms responsible for cavity noise, the current computational methods for cavity noise, and an experimental database for low Mach number flows in the open literature. Most of the conclusions have been drawn from reports for a gaseous medium (air) at moderate subsonic speeds and there may be limitations to directly applying these concepts and rules of thumb to a liquid medium (e.g. water) at lower subsonic speeds. A significant factor that affects cavity tones in water is the structural response of the cavity. The research in this thesis assumes that all walls are rigid. This work concentrates on in-air computations but future work should look at in-water calculations. This would require a modification to the solver to generalize the perfect gas law to allow an equation of state for water.

1.2.1 Cavity Noise Mechanisms

The mechanisms for cavity noise vary with the geometry of the cavity and are most often separated into two types: shallow and deep cavities. Detailed reviews of the physical mechanisms of cavity noise can be found in [5–7]. A recent review of cavity flows, albeit for higher Mach numbers, can be found in Reference [23]. The feedback mechanism for the two types of cavities is often controlled by different physical processes. Generally a length to depth ratio \(L/D\) ratio greater than one is called a shallow cavity and a \(L/D\) ratio less than one is a deep cavity [24]. The length, \(L\), is the opening in the streamwise direction and the depth, \(D\), is the distance from the cavity opening to the bottom of the cavity. The width of the cavity, \(W\), will also determine the three-dimensionality of the cavity. As the cavity width increases such that \(L/W\) is less than one it approximates a two-dimensional cavity. Also, as the cavity width increases, the spanwise correlations increase in the shear layer with both the broadband and tonal noise increasing [25]. The various dimensions for the cavity are found in Figure 1.2. Different physical processes often control the source mechanisms for the two types of cavities, although each
require a feedback loop and an oscillating shear layer.

![Diagram of cavity flow with dimensions L and D.]

Figure 1.2. Various dimensional parameters for cavity flows.

The noise produced from flow over cavities is broadband in nature but can include tones if the conditions are right. Most research concentrates on tones, since they are more offensive due to the extremely high sound pressure levels (SPL) that can occur. Accordingly, the research described in this thesis concentrates on tonal production. The oscillation frequency of the shear layer controls the frequency of vortex shedding, but vortex shedding from the upstream edge of the cavity is not a necessary condition for a cavity tone [6, 26]. However, in subsonic flows, the transformation of a shear layer to a train of vortices does provide a mechanism that limits the oscillation amplitudes, and hence, the cavity tone amplitudes [27].

The broadband noise often has a dramatically lower SPL and is generally of secondary concern. The broadband noise is harder to predict since it requires the resolution of the small scales of turbulence, which are difficult to simulate numerically. An example where broadband noise is of concern is the buffeting of passengers from an open car sun roof [9, 28]. The tendency of a cavity to generate a tone is independent of the turbulence in the flow field; tones may be generated for both laminar and turbulent flows [29]. While similar amplitude cavity tones can occur for laminar or turbulent flow the broadband noise is higher for turbulent flow over a cavity. This shows that the overall noise generated by cavity flow is dependent on turbulence.
1.2.1.1 Shallow Cavities

Shallow cavities have a $L/D$ ratio greater than one but at higher $L/D$ ratios the generation of cavity tones actually diminishes (although broadband noise is still an issue as the cavity becomes a combination of a backward and forward facing step). The geometry for a shallow cavity has a dramatic effect on the noise generated and can be used to classify the cavity as open, transitional, or closed, see Table 1.1 [30]. This nomenclature was developed for supersonic flows but has been applied to subsonic flows. An open cavity has a shear layer that spans the entire cavity and is responsible for the generation of most cavity tones. A closed cavity has a shear layer that attaches to the cavity floor, separates before reaching the rear wall of the cavity, and then reattaches downstream of the cavity. A closed cavity does not usually produce acoustic tones and can be modeled as a backward step followed by a forward step [30, 31]. The closed cavity has a larger $L/D$ ratio and the geometry is such that the feedback loop between the shear layer’s upstream separation region and the reattachment on the downstream corner is broken because the shear layer reattaches on the cavity floor before it reaches the downstream corner. Finally, a transitional cavity is one where the flow could be either open or closed. When the $L/D$ ratio is between eight and fourteen an experimental configuration could exhibit either open or closed behavior. Its behavior is influenced by many factors including geometric details or the unsteadiness of the freestream flow. The prediction of the precise behavior in this situation is very difficult because of the quasi-random response of the cavity.

<table>
<thead>
<tr>
<th>L/D Range</th>
<th>Type of Cavity</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 – 8</td>
<td>Open</td>
</tr>
<tr>
<td>8 – 14</td>
<td>Transitional</td>
</tr>
<tr>
<td>14+</td>
<td>Closed</td>
</tr>
</tbody>
</table>

The source mechanism for a shallow cavity is the shear layer with the frequency of interest either related to the longitudinal mode of the cavity or the instability frequency of the shear layer. As the flow passes the upstream cavity edge and separates, instability waves inherent in the shear layer cause it to oscillate. This shear layer then impinges on the downstream edge of the cavity, generating a
pressure pulse which travels upstream to the separation edge and “feeds back” energy to the separating shear layer. If the period of the shear layer oscillation is an integer multiple of the sum of the transit times for the instability wave to traverse the cavity and the acoustic wave to return, a feedback loop is established [27,32,33]. The generation of this cavity tone can also be modeled as a mass addition and expulsion at the downstream edge of the cavity, e.g. a pseudo-piston (See Figure 1.3). The pseudo-piston generates the pressure wave that completes the feedback loop. If a longitudinal mode of the cavity matches the shear layer frequency a strong tone is radiated to the far field. If the energy in the shear layer is high enough the cavity tone can be generated solely from the impinging shear layer.

![Figure 1.3. Pseudo-piston model for cavity tones.](image)

1.2.1.2 Deep Cavities

The cavity tones generated from flow over a deep cavity are due to a different mechanism than that for a shallow cavity. For a deep cavity the L/D is less than one and is classified as an open cavity, where the shear layer spans the entire cavity opening. The cavity tones generated are due to the same forcing function as a shallow cavity but the feedback loop is completed in a different manner. The primary method of noise generation for a deep cavity is a cavity response in the depth mode to an oscillating shear layer [34]. The shear layer separates from the upstream edge and oscillates due to instabilities in the shear layer, as in the shallow cavity. However, if the shear layer frequency matches a depth mode of the cavity
then the shear layer oscillation is reinforced and a strong tone is generated [29,34]. Here, the oscillating shear layer is the forcing function to drive the cavity. This process is different than in the shallow cavity where the pressure waves generated from the downstream edge complete the feedback loop. The deep cavity tone generation phenomenon is illustrated in Figure 1.4.

![Figure 1.4. Physical mechanism for a cavity tone for a deep cavity.](image)

An approximation for the radian frequency of a two dimensional cavity depth mode was derived by Tam for a Mach number of zero [35],

\[
\omega_{mn} = \pi \left[ \frac{(n - 1/2)^2}{(D/L)^2} + (m - 1)^2 \right]^{1/2}
\]

(1.1)

where \( m \) is the depth mode number and \( n \) is the longitudinal mode number. The effect of grazing flow over the cavity will slightly alter the depth mode frequency due to a change in the impedance of the cavity opening, but this should be negligible for very low Mach number flows. Experiments conducted by Block [2] for a Mach number between 0.05 and 0.4 measured the frequency response of deep cavities. The Strouhal number, a nondimensional frequency, was determined to be,

\[
S = \frac{fL}{U} = \frac{m}{1/\kappa + M \left( 0.7 + \frac{1.23}{L/D} \right)}
\]

(1.2)

where \( f \) is the frequency, \( U \) is the freestream speed, \( m \) is the depth mode number, \( M \) is the Mach number, and \( \kappa \) is an empirical constant, related to the convective wave speed of vortices, usually 0.57. Block showed that the dominant noise source for a cavity at a Mach number of 0.05 to 0.15 was a cavity responding in a depth
mode or Helmholtz mode.

Elder et al. [36] also conducted experiments in a low speed wind tunnel and found that for very low Mach numbers the cavity tone was an excitation of a normal pipe mode (depth response for a circular cavity). At these low Mach numbers the sound directivity pattern was omnidirectional. Elders [37] also developed a method to predict the frequency and radiated noise from turbulent flow over a cylindrical cavity. These experiments further indicate that a cavity tone for low Mach number flows is often due to a depth mode response. Tam and Block [33] experimentally showed that for Mach numbers below 0.2 all cavities, regardless of L/D ratio, respond in the depth mode. This may indicate that there is not enough energy transferred from the shear layer to complete the shallow cavity feedback loop.

1.2.2 Review of Theoretical Cavity Noise

At moderate subsonic Mach numbers the prediction of cavity tone frequencies is easily calculated through Rossiter’s formula [31]. The formulation relates the Strouhal number to the travel time a particle would take from the separation region at the upstream edge of the cavity to the downstream edge.

\[ S = \frac{f L}{U_0} = \frac{U}{L} \frac{m - \gamma}{m M + 1/\kappa}. \]  

(1.3)

Where \( m \) is the mode number (e.g. 1, 2, 3, ...), and is often equated with the number of vortices along the shear layer. \( L \) is the cavity length and \( \gamma = 0.25 \) is a constant that accounts the phase lag for the pressure wave to propagate from the downstream edge to the upstream edge and is slightly dependent on cavity depth [38]. \( \kappa \) is the convective velocity, normalized to the freestream velocity, of the vortex along the shear layer. A typical value is 0.57 [39] but this is slightly Mach number dependent. This formulation is also dependent on the cavity length to depth ratio, being primarily for shallow cavities \((1 < L/D < 4)\).

While this formulation is acceptable for moderate Mach numbers there is a correction for higher Mach numbers. Unfortunately, at low Mach numbers it no longer applies. Comparisons to experimental data show that the Mach number must be greater than 0.2 for this equation to hold. The breakdown of the formula at low Mach numbers is likely related to a change in convection velocity and the reduction of energy in the shear layer. Lastly, a seminal study in low Mach number
flows was conducted by Tam and Block [33, 40]. They measured the cavity tone frequencies and near field sound pressure levels for Mach numbers as low as 0.05. Hydrodynamic stability theory is used to provide an estimate of the cavity tone frequency. They proved that as the Mach number goes below 0.2 the cavity tone is generated by a different mechanism than at higher speeds. Regardless of the L/D ratio of the cavity, at low Mach numbers, if a cavity tone is sustained it is based on a depth mode of the cavity. This is further proof that Rossiter’s equation breaks down at low Mach numbers.

This equation is only for determining the frequency of the cavity tone. It is much more difficult to predict the amplitude of the tone since there are more variables that enter into it and it is a nonlinear process [41]. The instability waves of the shear layer are typically predicted with linear theory. However, the cavity resonance is a nonlinear process, which typically requires pressure or velocity data in the cavity to predict absolute amplitudes.

A more basic formulation for the frequency of standing waves in a cavity has been derived from room acoustics as,

\[ f = \frac{c}{2} \sqrt{\left(\frac{n_x}{L}\right)^2 + \left(\frac{n_y}{2D}\right)^2 + \left(\frac{n_z}{W}\right)^2} \]  

(1.4)

For the depth mode set \( n_x = n_y = 0 \) and then \( n_y = 0, 1, 3, 5, \ldots \). This has been appropriately modified to account for the open cavity in the \( +y \) direction.

Tam [35] has done significant work for two-dimensional, deep cavities at low Mach numbers. He provides a complex equation to determine the acoustic modes of cavities in a quiescent medium. A primary result of his work is a mathematical proof that there exists a minimum Mach number below which there is not sufficient energy in the shear layer to overcome the radiation damping of a cavity mode. Hence, at speeds below the minimum Mach number, cavity tones can not be self-sustained and do not occur. This holds true for deep cavities where the feedback loop is between the depth mode of the cavity and the shear layer. In shallow cavities the mechanism of feedback is between vortices in the shear layer and the creation of a pressure wave when the vortex impinges on the downstream cavity edge.

Work has been done by Elder and co-workers [36,37] on predicting the frequency and relative amplitude for low Mach number deep, cavities. Their work involves a
root-locus approach to solve the cavity feedback loop problem but is only applicable to cavities whose openings are small compared to the wavelength of the tone. This is different than the shallow cavity problem in that here the shear layer frequency matches the depth mode of the deep cavity, which is a circular pipe. The root-locus method uses a forward transfer function for the shear layer and a backward transfer function to model the cavity feedback. Where these two transfer functions intersect is the cavity tone frequency. To determine the relative amplitude of the cavity tone the root locus method also predicts the “Q” or sharpness of the tone. Elder provides a calibration constant to transform the Q to a relative near field pressure amplitude. Methodology is also presented to estimate the radiated noise from this near field pressure. Mast and Pierce [42] developed a very similar method to analyze cavity flow tones.

Howe has also done considerable work to understand very low Mach number cavity tones using idealized descriptions of vortex sheets, incompressible flow surrounding the cavity, and Green’s functions. Two of his papers [38, 43] concentrate on predicting the Strouhal frequency based on shear layer instabilities. His more recent paper [44] includes a method for the resonance frequency and the radiated sound produced by the flow over the cavity. He showed that at these low Mach numbers the sound radiation is proportional to the unsteady drag produced by the cavity flow. A directivity pattern for low Mach number cavity flows is also given.

1.2.3 Review of Low Mach Number Cavity Simulations

The use of CFD to predict cavity noise or pressure fluctuations at low Mach numbers represents only a small amount of the published information on cavity flows. This is because most of the work has been done for the aircraft industry, which operates at moderate subsonic Mach numbers. However, there is a growing number of research work being conducted for the automobile industry. Up until recently the research that has been conducted at low Mach number mostly falls within three disciplines: DNS, perturbation, and hybrid. CHOPA represents a non-hybrid numerical tool with highly accurate spatial discretizations.

One of the more prevalent DNS simulations is that by Colonius, Basu, and Rowley [14] of low Mach number cavity flows was of two-dimensional laminar cavities that varied the L/D ratio between one and five and an inflow Mach number
between 0.2 and 0.95. The flow simulation was conducted using DNS to examine the existence of a “wake-mode” in cavity flow, which is where the cavity oscillates violently due to large eddies similar in dimension to the cavity. The wake mode has only been identified for two-dimensional problems [1]. Fluctuating pressures and visualizations of the flow within the cavity were presented. Chang and Park [45] also perform cavity simulations at a Mach number of 0.1 using an incompressible Navier-Stokes solver. However, they did not calculate any acoustic information, only the velocity field. With recent improvements in computing power there are more researchers using a CAA (Computational Aero Acoustics) approach. Lee et al. [46] performed both DNS and LES (Large Eddy Simulations) calculations for deep and shallow cavity geometries. A DNS study on the stability of open cavity flows using DNS was conducted by Vinha et al. [47].

The perturbation approach uses an Expansion about Incompressible Flow (EIF) technique where an acoustic solver takes the flow variables from an incompressible flow solver to calculate the flow induced noise. The work by Hardin and Pope [48] used the EIF technique to calculate the flow over a cavity (M = 0.1) with a \( L/D \) of four. The fluctuating pressure was plotted at a variety of angles but there was no comparison to validation data. The work of Slimon et al. [49] expanded the EIF technique and conducted several benchmark cases, in addition to a cavity noise case. The cavity noise simulation was for an inflow Mach number of 0.26 and a cavity \( L/D \) of 0.5. Comparisons were made to the cavity tone frequencies predicted from Rossiter’s equation, with good results. Another use of the EIF approach is one used by Moon et al. [50]. For their low Mach number simulations they use an incompressible Navier-Stokes solver for the hydrodynamic flow field then a hydrodynamic/acoustic splitting technique to couple the flow and acoustic field. They were able to accurately simulate cavity flow tones at Mach numbers as low as 0.08.

Hybrid codes that use a combination of a fluid dynamic solver with an acoustic solver are becoming more prevalent. Ashcroft et al. use the NASA designed CFD code CFL3D [51], which is a thin layer Navier-Stokes solver, to simulate a Mach number 0.15 cavity flow representing an automobile door gap. While they use a Ffowcs Williams-Hawkins method, most of the comparison are to wall pressures in the cavity, due to the available experimental results. This is a good paper that highlights the difficulties in obtaining far field cavity tones. Hao et al. used
a LES solver with Lighthill’s acoustic analogy to predict flow induced noise for cavities for low Mach number flow [16]. The wall pressure fluctuations and near field noise were predicted with their tool. Low Mach number cavity flows were also predicted using a hybrid method by Yenigelen and Zafer [52]. The results of the fluid dynamic solver were used with a Modified Curle’s analogy to predict the far field noise generated by flow over shallow cavities with $L/D = 2$.

With the increase in computing power more computationally intensive solution methods have become more prevalent. Numerous other researchers have examined cavity flows using LES to predict the cavity flow and resultant noise. These did not utilize preconditioning so there are difficulties in solving for low Mach number cavity flows. Bagell used a LES simulation to characterize the flow over a deep cavity [13]. Automobile flows have received a lot of attention from researchers, primarily to understand the effect of buffeting on open sunroofs [53–55]. Each of these researchers used different LES solvers to understand the impact of open windows or sunroofs on passenger comfort.

CRAFT Tech has performed numerical cavity simulations for both supersonic cavities for weapons bays [56, 57] and very low Mach number cavities for submarines [11, 11, 58]. They used their CFD code CRUNCH, which is an unstructured flow solver that uses preconditioning to efficiently solve low Mach number flows. However, they do not calculate the radiated noise for the cavities and only compare the cavity wall pressure fluctuations to experimental data to understand resonant and non-resonant conditions. They use a hybrid RANS/LES turbulence model to accurately model the shear layer, which is an important part of the cavity feedback loop.

Similar to the cavity flows of interest for this report, and using a full CAA method, is the work by Koh and Moon [59]. They use a $6^{th}$ order accurate method for the spatial discretization and simulate cavity flows from Mach numbers of 0.1 to 0.4. An explicit Reynolds Algebraic Stress Model is used for the turbulence quantities. While they predict the tonal components well for these cases, the cavities of interest in the present research are at a lower Mach number and require the use a preconditioned method for their efficient solution, something that Koh and Moon do not utilize. Conversely, Fiedler et al. solved for a cavity flow in turbomachinery and use a low Mach number preconditioner [60]. However, this is a CFD solver instead of a CAA solver and only uses a lower accuracy for the
spatial stencils.

1.2.4 Review of Low Mach Number Cavity Experiments

The validation data (e.g. frequency and amplitude of cavity tone) for low Mach number cavity flows is fairly limited in the public domain. Most of industry’s needs have been in moderate subsonic flows and greater, due to military and civilian aircraft requirements. There is a wide range of measurements made for cavity flows, with no single experiment having all the quantities of interest to fully validate a numerical tool. As such, several different comparisons to experimental data are required. The flow features of the cavity may be validated with one set of data while the acoustic tones are validated with another. The low Mach number experimental database that has been reviewed is found in Table 1.2.
Table 1.2. Experimental database for low Mach number cavity flows.

<table>
<thead>
<tr>
<th>Type of Data</th>
<th>Type of Cavity</th>
<th>Turbulent/Laminar</th>
<th>Mach Number</th>
<th>Ref.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency vs. L</td>
<td>Deep</td>
<td>Turbulent</td>
<td>0.4</td>
<td>61</td>
</tr>
<tr>
<td>Flow visualization</td>
<td>Shallow</td>
<td>Turbulent</td>
<td>0.01</td>
<td>62</td>
</tr>
<tr>
<td>Flow visualization</td>
<td>Shallow</td>
<td>Turbulent</td>
<td>0.03</td>
<td>63</td>
</tr>
<tr>
<td>Wall pressure</td>
<td>Shallow</td>
<td>Turbulent</td>
<td>0.05</td>
<td>64</td>
</tr>
<tr>
<td>Flow visualization</td>
<td>Shallow</td>
<td>Turbulent</td>
<td>&lt; 0.001</td>
<td>65</td>
</tr>
<tr>
<td>Wall pressure</td>
<td>Deep</td>
<td>Turbulent</td>
<td>0.1</td>
<td>66</td>
</tr>
<tr>
<td>Strouhal vs. D, Empirical depth mode response</td>
<td>Deep</td>
<td>Turbulent</td>
<td>&lt; 0.18</td>
<td>24</td>
</tr>
<tr>
<td>SPL, directivity</td>
<td>Deep/Shallow</td>
<td>Turbulent</td>
<td>0.1 - 0.25</td>
<td>67</td>
</tr>
<tr>
<td>Velocity profile, Strouhal number</td>
<td>Shallow</td>
<td>Laminar</td>
<td>0 - 0.08</td>
<td>26</td>
</tr>
<tr>
<td>Various</td>
<td>Deep/Shallow</td>
<td>Both</td>
<td>Various</td>
<td>5</td>
</tr>
<tr>
<td>Strouhal vs. Mach number, L/D, L</td>
<td>Deep/Shallow</td>
<td>Turbulent</td>
<td>0.05 - 0.4</td>
<td>2</td>
</tr>
<tr>
<td>Frequency, Wall pressure, Boundary Layer</td>
<td>Deep/Helmholtz</td>
<td>Turbulent</td>
<td>0.02 - 0.07</td>
<td>68</td>
</tr>
<tr>
<td>Frequency</td>
<td>Deep</td>
<td>Turbulent</td>
<td>0.05</td>
<td>[37]</td>
</tr>
<tr>
<td>Flow visualization, frequency vs. velocity</td>
<td>Deep</td>
<td>Turbulent</td>
<td>&lt; 0.15</td>
<td>[69]</td>
</tr>
<tr>
<td>Turbulent Intensity profile, Velocity profile, Frequency, Directivity</td>
<td>Deep</td>
<td>Laminar/Turbulent</td>
<td>&lt; 0.2</td>
<td>70</td>
</tr>
<tr>
<td>Flow visualization</td>
<td>Shallow</td>
<td>Turbulent</td>
<td>0.1 - 0.2</td>
<td>71</td>
</tr>
<tr>
<td>Frequency</td>
<td>Deep</td>
<td>Turbulent</td>
<td>0.09</td>
<td>72</td>
</tr>
<tr>
<td>Frequency vs. length, flow visualization, velocity profiles, Cp on cavity walls, Reynolds Stress on cavity walls,</td>
<td>Deep/Shallow</td>
<td>Laminar</td>
<td>0.001</td>
<td>4</td>
</tr>
<tr>
<td>Flow visualization, velocity profiles</td>
<td>Deep/Shallow</td>
<td>Laminar</td>
<td>0.001</td>
<td>73</td>
</tr>
<tr>
<td>Spectral plots (arbitrary level), Frequency</td>
<td>Deep</td>
<td>Turbulent</td>
<td>0.001</td>
<td>36</td>
</tr>
<tr>
<td>Wall pressures, Velocity Profile, Frequency vs. speed, SPL vs. flow rate</td>
<td>Deep</td>
<td>Turbulent</td>
<td>&lt; 0.1</td>
<td>74</td>
</tr>
<tr>
<td>Wall pressures, SPL, Directivity</td>
<td>Shallow</td>
<td>Turbulent</td>
<td>0.12 - 0.25</td>
<td>76</td>
</tr>
<tr>
<td>Frequency vs. Speed, Spectra</td>
<td>Shallow/Deep</td>
<td>Turbulent</td>
<td>0.1 - 0.5</td>
<td>40</td>
</tr>
<tr>
<td>Frequency vs. Near field pressure</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Frequency vs. Speed</td>
<td>Shallow</td>
<td>Turbulent</td>
<td>0.1 - 0.4</td>
<td>77</td>
</tr>
<tr>
<td>Frequency vs. speed</td>
<td>Deep</td>
<td>Turbulent</td>
<td>&lt; 0.03</td>
<td>10</td>
</tr>
<tr>
<td>SPL vs. cavity geometry, Frequency vs. speed</td>
<td>Deep</td>
<td>Turbulent</td>
<td>0.12 - 0.24</td>
<td>78</td>
</tr>
<tr>
<td>Velocity profiles, far field SPL</td>
<td>Shallow</td>
<td>Turbulent</td>
<td>0.18 - 0.4</td>
<td>79</td>
</tr>
<tr>
<td>Frequency vs. Speed</td>
<td>Shallow</td>
<td>Turbulent</td>
<td>0.1 - 0.3</td>
<td>80</td>
</tr>
<tr>
<td>Cp on cavity walls vs. Mach number</td>
<td>Shallow</td>
<td>Turbulent</td>
<td>0.2 - 0.95</td>
<td>30</td>
</tr>
<tr>
<td>Boundary layer spectrum</td>
<td>Shallow/Deep</td>
<td>Turbulent</td>
<td>0.035</td>
<td>81</td>
</tr>
</tbody>
</table>
1.3 Objective of Current Study

The main objective of this work is to accurately predict the frequency and amplitude of cavity tones for very low Mach number hydrodynamic flows. A computational aeroacoustical (CAA) approach is used to simulate both the hydrodynamic and acoustic field surrounding the cavity.

The importance of this work is directly applicable to the automobile and naval industries. For open sun roofs, the mitigation of pressure fluctuations increases the comfort of passengers, while underwater vehicles require a low acoustic signature to effectively carry out their missions. In extreme cases the large pressure fluctuations in cavities can cause structural fatigue. A better understanding of the mechanisms of cavity noise can also help in the design of underwater vehicles while still allowing them to meet acoustic requirements. This code could also be applied to aircraft in approach configurations.

In the next chapter the underlying mathematical principles of the CAA solver will be discussed. The application of preconditioning to the solver will be described as well as a wide range of preconditioners. A discussion of the modifications needed to enable preconditioning are also given as well as the implementation of the boundary conditions. The theory behind the Ffowcs Williams-Hawkings far field propagation method will be given. This is a very effective method to post process the near field acoustical data to provide radiated noise predictions at a fraction of the computing cost.

The third chapter provides various validation cases for the numerical code. Although the previous version of CHOPA (Compressible, High-Order, Parallel Acoustics) has been tested extensively for moderate subsonic cavity and jet flows, the numerous modifications made to the code during this work necessitate the need for additional validation of the code. To that end, simple validation problems, such as the convection of velocity perturbations out of the domain will estimate the convergence improvement from preconditioning. Likewise, the boundary conditions will be validated.

In Chapter 4 the low Mach number flow over a two-dimensional cavities is presented. Both a laminar and a turbulent cavity are simulated and the near field pressure and velocities are compared to experimental values. The following chapter will extend the results to both laminar and turbulent cavities in three
dimensions. The simulations are much more computationally intensive but, with preconditioning, can be run fast enough to provide engineering judgement in real time designs.

Lastly, in Chapter 5 general conclusions are made about the low Mach number solver and the effectiveness of preconditioning. Possible avenues for future research are also provided.
Chapter 2  |  Mathematical Formulation

This chapter describes the governing equations used for the low Mach number cavity simulations. The background behind the preconditioning as well as a discussion of various preconditioners is given. Boundary conditions and artificial dissipation methods specific for low Mach number flows are discussed and the theoretical formulation behind the far field propagation method is described.

2.1 The Governing Equations

The fluid flow past a cavity can be analyzed with the compressible Navier-Stokes equations. The derivation of the Navier-Stokes can be found in numerous textbooks [82, 83] so only the equations are presented here. A brief description of the numerical methodology behind the solution of the equations is presented first.

The Navier-Stokes equations can be solved with a numerical method called DNS (Direct Numerical Simulation) and no approximations are required to fully resolve the flow. Due to the range of scales present in a turbulent flow the number of grid points required to accurately resolve the flow is proportional to $Re^{9/4}$. Typically, simplifications are made that model the same range of scales of the turbulent fluctuations. The Compressible, High-Order, Parallel, Acoustic (CHOPA) code is based on a short-time Reynolds averaging and Favre averaging of the governing equations. A more complete derivation of the governing equations is given by Shieh [12]. Short time Reynolds averaging is quite common and is found in URANS (Unsteady Reynolds Averaged Navier-Stokes) solvers. By also performing a short-time Favre averaging of the instantaneous equations the compressible equations are simplified. Usually this type of averaging is only required to effectively
simulate high subsonic flows but since the base version of CHOPA includes it, it
does not affect adversely low Mach number solutions and it is kept.

Since the CHOPA numerical tool is a CAA code there are a slight difference
in the nondimensionalization parameters. The normalizing reference flow velocity
used is the speed of sound. The Navier-Stokes equations are nondimensionalized
in the following way, where an asterisk denotes a dimensional quantity.

\[ x_i = \frac{x_i^*}{L_r^*}, \quad u_i = \frac{u_i^*}{c_\infty^*} \]  
\[ \rho = \frac{\rho^*}{\rho_\infty^*}, \quad p = \frac{p^*}{\rho_\infty^* c_\infty^*} \]  
\[ T = \frac{T^*}{T_\infty^*}, \quad t = \frac{t^*}{L_r^*/c_\infty^*} \]  
\[ \mu = \frac{\mu^*}{\mu_\infty^*}, \quad \tau_{ij} = \frac{\tau_{ij}^* L_r^*}{\mu_\infty^* c_\infty^*} \]  

The subscript \( \infty \) refers to a freestream value and \( L_r^* \) is a reference length (e.g. cavity length).

An equation of state is used to relate the thermodynamic variables and provide
closure to the system of equations. An ideal gas is assumed in this work, but other
equations of state (including one for water) are available. Thus, the nondimensional
perfect gas law is,

\[ p = \frac{\rho T}{\gamma} \]  

where \( \gamma = c_p/c_v \) and is the ratio of specific heats.

The short-time Reynolds and Favre averaging processes are defined as,

\[ \bar{q} = \lim_{T \to \infty} \frac{1}{T} \int_{t_0}^{t_0+T} q \, dt \]  
\[ \bar{e}q = \lim_{T \to \infty} \frac{1}{T} \int_{t_0}^{t_0+T} eq \, dt \]  

Using those definitions the variables are decomposed as,
\[ u_i = \bar{u}_i + u''_i, \quad T = \bar{T} + T'' \quad (2.8) \]
\[ e = \bar{e} + e'' , \quad \tau_{ij} = \bar{\tau}_{ij} + \tau''_{ij} \quad (2.9) \]
\[ \rho = \bar{\rho} + \rho' , \quad p = \bar{p} + p' \quad (2.10) \]
\[ q_j = q_L_j + q'_j. \quad (2.11) \]

In the decomposition an overbar, denotes the time-dependent average value, a prime denotes a fluctuation about the time average, a tilde is the mass-averaged value, and a double prime denotes the fluctuation about the mass average. Insertion of these averaged variables into the Navier-Stokes equations and the equation of state results in the short-time Favre averaged equations. This gives additional terms that must be modeled through various closure options (See [12]).

The first closure is a model for the Favre-averaged Reynolds stress tensor that is modeled using the Boussinesq eddy viscosity approximation [84] given by Equation (2.12). As the code is written for an ideal gas the bulk viscosity is related to the molecular viscosity through Stokes Hypothesis, Equation (2.13). Stokes Hypothesis is generally valid for gases in most common CFD solvers (e.g. air). However, the coefficient of bulk viscosity cannot be assumed negligible for gases when there are shock waves or attenuation and absorption of acoustic waves [83]. Since the CHOPA code is designed with the intent of predicting acoustic waves the validity for Stokes Hypothesis is examined further. The frequency content of the anticipated cavity tones for this thesis is expected to be in the kHz range. A comparison of the corresponding wavelengths for the acoustic tones well exceeds the mean free path of the gas molecules. The wavelength being much greater than the molecular free path of the gas molecules permits the use of Stokes Hypothesis. For clarity the molecular or laminar stress tensor is given in Equation (2.14).

\[ \{ar{\rho}\bar{\tau}_{ij}\} = -\rho u_i'' u_j'' = \frac{M}{\text{Re} 2\mu_t} \left[ \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{1}{3} \frac{\partial \bar{u}_k}{\partial x_k} \delta_{ij} \right] \quad (2.12) \]
\[ \mu_B = -\frac{2}{3} \mu. \quad (2.13) \]
\[\tau_{ij} = \frac{M}{Re} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \mu_B \frac{\partial u_k}{\partial x_k} \delta_{ij} \right] \]  \hspace{1cm} (2.14)

Where \( \mu_t \) is the turbulent eddy viscosity, \( M = \frac{U^*}{c_\infty} \), \( Re = \frac{\rho U^* L_r}{\mu_\infty} \), \( \mu_B \) is the bulk viscosity, \( \mu_t \) is the turbulent viscosity, and \( \delta_{ij} \) is the Kronecker delta function. As mentioned before, Stokes Hypothesis is only valid for gases in some situations. If a change to a more general solver that is applicable to both water and air is made then the Stokes hypothesis is not valid and a different approximation is required. The molecular viscosity, \( \mu \), is related to temperature through Sutherland's formula. This is valid only for ideal gases [22]. In nondimensional form the viscosity is given by,

\[ \mu = \left( 1 + \frac{C_2}{T^*_\infty} \right) \left( \frac{T^{*1.5}}{T^* + C_2/T^*_\infty} \right) \]  \hspace{1cm} (2.15)

where, \( T^*_\infty \) is the freestream temperature and \( C_2 \) is 110.4 K for air. Note that this is the dimensional form.

The other closure involves the heat transfer term, \( q_j \). This term models heat conduction through Fourier’s Law and is given by,

\[ q_j = -\frac{1}{\gamma - 1} \frac{M \mu}{Re \ PR_L} \left( \frac{\partial T}{\partial x_j} \right) \]  \hspace{1cm} (2.16)

Where, \( PR_L \) is the laminar Prandtl number, \( c_p \mu_\infty / \kappa \), and is approximately 0.72 for air. For the turbulent heat transfer, given below, \( PR_T \) is the turbulent Prandtl number and is assumed to be 0.9. This value is assumed constant because the flow is low speed with minimal heat transfer. The corresponding turbulent heat transfer formulation is given in Equation (2.17).

\[ q_{Tj} = \rho u_j e' = -\frac{1}{\gamma - 1} \frac{M \mu_t}{Re \ PR_T} \left( \frac{\partial \tilde{T}}{\partial x_j} \right) \]  \hspace{1cm} (2.17)

Finally, if a Newtonian fluid is assumed, and applying the applicable closure approximations, then the nondimensional Navier-Stokes equations can be written in tensor notation as,

\[ \frac{\partial \tilde{\rho}}{\partial t} + \frac{\partial \tilde{\rho} \tilde{u}_i}{\partial x_i} = 0 \]  \hspace{1cm} (2.18)
\[
\frac{\partial (\bar{\rho} \tilde{u}_i)}{\partial t} + \frac{\partial }{\partial x_j}(\bar{\rho} \tilde{u}_j \tilde{u}_i) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial }{\partial x_j}(\bar{\rho} \tilde{\tau}_{ij} + \{\bar{\rho} \tilde{\tau}_{ij}\}) \tag{2.19}
\]

\[
\frac{\partial (\bar{\rho} \tilde{E}_i)}{\partial t} + \frac{\partial }{\partial x_j} \left[(\bar{\rho} \tilde{E}_j + \bar{p}) \tilde{u}_j\right] = -\frac{\partial }{\partial x_j} \left(q_{Lj} + q_{Tj}\right) + \frac{\partial }{\partial x_j} \left[\tilde{u}_i (\tilde{\tau}_{ij} + \{\bar{\rho} \tilde{\tau}_{ij}\})\right] \tag{2.20}
\]

\[
\bar{p} = \frac{\bar{p} \tilde{T}}{\gamma} \tag{2.21}
\]

Remember that the terms in curly brackets are modeled with a turbulence model. An advantage of this averaging process, when coupled with a turbulence model based on the Boussinesq approximation (e.g. the Spalart-Allmaras turbulence model [85]) is that the equations have the same form as the laminar Navier-Stokes equations. The exception is that the viscosity coefficient is replaced with \((\mu + \mu_T)\) and the heat transfer terms are replaced by \((\mu/Pr + \mu/Pr_T)\). For the remainder of the thesis the averaging and fluctuation symbols will be omitted for simplicity.

As such, the Navier-Stokes equations can be written in vector form as,

\[
\frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial \mathbf{E} - \mathbf{E}_v}{\partial x} + \frac{\partial \mathbf{F} - \mathbf{F}_v}{\partial y} + \frac{\partial \mathbf{G} - \mathbf{G}_v}{\partial z} = \mathbf{H} \tag{2.22}
\]

\[
\mathbf{Q} = \begin{cases}
\rho \\
\rho u \\
\rho v \\
\rho w \\
\rho E_t
\end{cases}, \quad (2.23)
\]

\[
\mathbf{E} = \begin{cases}
\rho u \\
\rho u^2 + p' \\
\rho uv \\
\rho uw \\
(\rho E_t + p) u
\end{cases}, \quad \mathbf{E}_v = \begin{cases}
0 \\
\tau_{xx} \\
\tau_{xy} \\
\tau_{xz} \\
\nu \tau_{xx} + \nu \tau_{xy} + \nu \tau_{xz} + q_x
\end{cases}, \tag{2.24}
\]
\[
\mathbf{F} = \begin{Bmatrix}
\rho v \\
\rho w \\
\rho w v + p' \\
\rho v w \\
(\rho E_t + p) v
\end{Bmatrix}, \quad \mathbf{F}_v = \begin{Bmatrix}
0 \\
\tau_{xy} \\
\tau_{yy} \\
\tau_{yz} \\
u \tau_{xy} + v \tau_{yy} + w \tau_{yz} + q_y
\end{Bmatrix}, \quad (2.25)
\]

\[
\mathbf{G} = \begin{Bmatrix}
\rho w \\
\rho u w \\
\rho u w v + p' \\
\rho v w \\
(\rho E_t + p) w
\end{Bmatrix}, \quad \mathbf{G}_v = \begin{Bmatrix}
0 \\
\tau_{xz} \\
\tau_{yz} \\
\tau_{zz} \\
u \tau_{xz} + v \tau_{yz} + w \tau_{zz} + q_z
\end{Bmatrix}, \quad (2.26)
\]

Where a subscript indicates a derivative with respect to that subscript. The viscous terms have a \( v \) subscript and the \( \mathbf{H} \) vector contains any source terms. A perturbation pressure is defined by \( p' \) and is a gauge pressure\(^1\). This is used to help prevent errors associated with numerical roundoff. Also, \( E_t \) is the total energy defined in Cartesian tensor notation as,

\[E_t = \frac{T}{\gamma (\gamma - 1)} + \frac{1}{2} u_i u_i \quad (2.27)\]

### 2.2 Numerical Discretization

To solve the computational aeroacoustic problem the Navier-Stokes equations are discretized in time and space using a finite difference formulation. This section describes the methods used in this research.

#### 2.2.1 Spatial Discretization

Computational aeroacoustic (CAA) problems are similar to CFD methods except the discretization methods are typically of a higher order of accuracy to resolve the small perturbations about the mean flow associated with the radiated sound. A high order numerical scheme is not the only sufficient requirement to accurately

\(^1\)This is based on the assumption that the mean static pressure is constant.
resolve the pressure wave. As such the Dispersion Relation Preserving (DRP) method, developed by Tam and Webb [86] and widely used for CAA simulations, is used for the spatial discretization. Since its original development, others [87,88] have optimized the method for curvilinear grids but the original formulation is adopted here.

The benefit of the DRP scheme for simulating an acoustical problem is that the dispersion relation is the same for both the finite difference scheme as well as the original partial differential equations over a prescribed wavelength range. Typically, the equation for a partial derivative is expanded in a Taylor series and then the coefficients are determined by equating like powers of $\Delta x$. However, in the DRP scheme those coefficients are determined by requiring the Fourier transform of the finite difference equation to be a close approximation of the Fourier transform of the exact spatial derivative. An integrated error is then used, based on the wavenumber range of interest for this problem, to optimize the coefficients. A more detailed description is given by Paul [89]. A seven point stencil is used in this work, the coefficients of which can be found in Appendix B.

### 2.2.2 Time Discretization

The time integration employs a dual time-step method for time accurate flows, which uses both an implicit and explicit method. The dual time-step framework is needed to allow enhanced steady-state solution methods to be applied to time accurate flows. These convergence acceleration methods include preconditioning and multigrid. More detailed information on the time accurate integration method is given by Shieh [12]. The next sections will describe the time integration methods utilized in the code.

#### 2.2.2.1 Dual Time-Step

Preconditioners are proven methods to accelerate the convergence of fluid dynamic flows but cannot handle unsteady flows [1,90–92]. The preconditioner modifies the eigenvalues of the governing equations and increases the convergence rate. These modifications destroy the time accuracy of the numerical method by equalizing the acoustic and convective wave speeds. However, a technique known as dual time-stepping can be used to recapture the time accuracy of the problem [91,93]. This
method can be used with any number of convergence acceleration techniques and is not limited to just preconditioning. At each pseudo-time step (e.g. inner loop) preconditioning is used to converge the solution rapidly. Once a converged steady state solution is obtained, the pseudo-time solution becomes part of the solution used to obtain the next physical time step solution (e.g. outer loop). Thus, the solution is broken into a series of steady-state solutions that are connected by outer loop of the dual time step scheme, see Figure 2.1. A side effect of a dual time-step method is that the physical time derivative behaves like a sink term during the inner loop, increasing the amount of damping [94]. This may necessitate a decrease in the amount of artificial dissipation or the solution may damp important high-wavenumber modes.

Figure 2.1. Interdependency of physical time (t) and pseudo-time (τ) in a dual time-stepping scheme.

This method will be illustrated for the Euler equations, but it is straightforward to extend it to the Navier-Stokes equations. The time dependent Euler equations can be written in vector form as,

$$\frac{\partial Q}{\partial t} = - \left( \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} \right) = -R(Q)$$

(2.28)

where $R$ is the residual. This residual includes the convective and diffusive fluxes as well as the artificial dissipation. By introducing a fictitious time, $\tau$, the dual time system of equations is given by,

$$\frac{\partial Q}{\partial \tau} = - \left[ \frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} \right] = -R^*(Q).$$

(2.29)

A preconditioner can be used to quickly drive this new residual to zero. The new system can be written as,

$$\Gamma \frac{\partial Q}{\partial \tau} + R^*(Q) = 0$$

(2.30)
where \( \Gamma \) is the preconditioner matrix. The final system given by,

\[
\Gamma \frac{\partial Q}{\partial \tau} + \left[ \frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} \right] = 0.
\] (2.31)

At steady state, the artificial time derivative is zero and the original equations are recovered. By using a dual time-step method, different time integration methods can be tailored to the inner, non-physical time loop, as well as the outer, physical time loop. Weiss and Smith [90] provide a good methodology to implement a dual time-step method with a Runge-Kutta method in pseudo-time and a backward implicit time integration method in physical time. Alves [95] and Falcao et al. [96] developed a Runge-Kutta time integration scheme to specifically take advantage of low Mach number preconditioning.

### 2.2.2.2 Explicit Time Integration

The heart of the time integration method is based on a modified four-step Runge-Kutta scheme, originally proposed by Jameson [97]. As an additional feature the code can be run in a purely time accurate mode that uses only the Runge-Kutta method. When the Navier-Stokes equations in computational space are written in vector form (see Appendix A) the solution vector at each intermediate step, as well as the next time step \( (n+1) \) is given as,

\[
Q^{(0)} = Q^n
\] (2.32)
\[
Q^{(1)} = Q^{(0)} - \alpha_1 J \Delta t \left[ \mathcal{R} \left( Q^{(0)} \right) \right]
\] (2.33)
\[
Q^{(2)} = Q^{(0)} - \alpha_2 J \Delta t \left[ \mathcal{R} \left( Q^{(1)} \right) \right]
\] (2.34)
\[
Q^{(3)} = Q^{(0)} - \alpha_3 J \Delta t \left[ \mathcal{R} \left( Q^{(2)} \right) \right]
\] (2.35)
\[
Q^{(4)} = Q^{(0)} - \alpha_4 J \Delta t \left[ \mathcal{R} \left( Q^{(3)} \right) \right]
\] (2.36)
\[
Q^{n+1} = Q^{(4)}
\] (2.37)

In these equations the superscripts refer to the timestep, \( n \), or an intermediate
step. The coefficients for integration, $\alpha$, are $\alpha = [0.5, 0.375, 1.0, 1.0]$ and are based on damping characteristics that are suitable for CAA simulations [98]. It should be noted that $\alpha_1$ and $\alpha_2$ are transposed from the original reference as the initial method was unstable. $J$ is the Jacobian of the transformation used to represent the physical grid in computational space. Additionally $\mathcal{R}$ is the residual at that intermediate step and is given by,

$$\mathcal{R} = \frac{\partial E}{\partial \xi} + \frac{\partial F}{\partial \eta} + \frac{\partial G}{\partial \zeta} - D(Q^{(0)})$$

$D$ is the selective artificial dissipation, which is calculated at the first intermediate step and is frozen for the remainder of the time step.

The modified Runge-Kutta method described above is formally fourth order accurate if the governing equations are constant coefficient partial differential equations. When that does not hold true it is second order accurate. The original Runge-Kutta method [22] is fourth order accurate regardless of the type of partial differential equation.

The reason the potentially lower order accurate integration method is selected is to reduce the memory requirements of the simulation. The original method requires the residual to be kept for all intermediate time steps, while a running summation can be used for the modified method. This is not a concern for two-dimensional problems but the memory constraints are substantial for three dimensions. Thus, the modified Runge-Kutta method is selected. If problems arise that require a higher accuracy in time, the original Runge-Kutta method can be implemented with relative ease.

### 2.2.2.3 Implicit Time Integration

In a dual time-step formulation the time accurate solution is broken down into a series of steady-state computations that comprise the unsteady solution. Each steady-state solution is solved via the modified Runge-Kutta method discussed above. The unsteady solution is calculated via an implicit backward method. Examining the fictitious residual in Equation (2.30) the physical time derivatives are discretized with a three-point backward differencing formula,

$$\Gamma \frac{\partial Q}{\partial \tau} = \frac{3Q^{n+1} - 4Q^n + Q^{n-1}}{2\Delta t} + \mathcal{R}(Q^{n+1}) = \mathcal{R}^*(Q^{n+1}).$$  (2.40)
This is a second order accurate linear implicit multistep method. In this equation \( n \) signifies a time step. To arrive at each physical time step, \( n \), the modified Runge-Kutta method is used with the preconditioned equation set until a convergence criteria is reached. Then the solution at the next physical time is calculated from Equation (2.40).

As this implicit method requires a minimum of three time steps it is called a non-starting method. Thus, a time accurate Runge-Kutta solution is used to calculate the first three time steps, then the dual time-step method is initiated. Additionally, the solution at each physical time step is initialized \( (Q^*) \) based on a three-point backward formula that is the estimate of solution variables at the next time step.

\[
Q^* = Q^n + \frac{3Q^n - 4Q^{n-1} + Q^{n-2}}{2}
\]  

(2.41)

A key concern for this time integration method is the stability of the method. Instabilities in the numerical method can occur when the fictitious time step, \( \Delta \tau \), exceeds the physical time step, \( \Delta t \). A detailed stability analysis can be found in Shieh [12]. Based on that analysis and the work of Arnone [99] for multigrid methods the fictitious time step is modified such that,

\[
\Delta \tau = \min \left[ \Delta \tau, \frac{\Delta t}{2^{l-1}(1/2)(1/CFL)} \right]
\]  

(2.42)

Here CFL is the Courant-Friedrich-Levy number and a used stability criterion. The number of multigrid levels being used is given by \( l \).

### 2.2.2.4 Calculation of Time Step

In a numerical discretization scheme the time step (\( \Delta t \)) is used to advance the solution in time\(^2\). For a steady-state solution a local time step can be used, where each point in the flow can be advanced in time with a different \( \Delta t \). In a time accurate solution the time step is global, where the minimum time step throughout the domain must be used to advance all points in time. For an explicit scheme, the minimum time step is inversely related to the characteristics of the flow. The time step required based on the acoustic eigenvalue is much smaller than the time

\(^2\)The discussion herein applies to the steady state solution (e.g. inner loop)
step based on the convective eigenvalue. This results in a very small time step to
convect particles, leading to excessively long run times. This is the primary reason
a preconditioner is adopted for the low Mach number solution\(^3\). In addition to
advancing the solution the time step also contributes to the effectiveness of the
preconditioner, as the viscous portion of the preconditioner is related to the time
step. The selection of a time step is based on the type of flow: inviscid or viscous.

The high aspect ratio of grid cells and the diffusivity in viscous flows necessitate
two different preconditioning methods. The mathematical basis for an inviscid time
step is well documented from first principles and will be discussed first. The time
integration method used here is an explicit method so the stability is governed
by the CFL (Courant-Friedrich Lewy) number. In a one-dimensional, Cartesian
coordinate system the CFL number is,

\[
\text{CFL} = \frac{\lambda}{\Delta x}.
\] (2.43)

Where \(\lambda\) is the maximum (i.e. acoustic = \(u+c\)) eigenvalue. This can be extended to
a three-dimensional, generic coordinate system by summing each directional acous-
tic eigenvalue and selecting the minimum grid spacing. Since the CAA calculation
is performed in computational space on a uniform grid, which is a transformation
of the original grid based on a Jacobian, the spatial distances are typically unity
and drop out of the formulation. Rearranging and solving for the inviscid time
step results in,

\[
\Delta t_i = \frac{\text{CFL}}{\lambda_c + \lambda_\eta + \lambda_\zeta}.
\] (2.44)

While the inviscid time step formulation is based on strict numerical stabil-
ity, the viscous time step formulation is somewhat less well defined. The viscous
time step (used in this research) is from Swanson and Turkel’s report [100], which
was based on Muller’s [101] work on numerical stability of the linearized Navier-
Stokes equations. The final result is formulated for CHOPA and has been success-
fully verified against a formulation available in NASA’s public domain CFD code,
CFL3D [102].

The viscous time step uses the concept of viscous eigenvalues, similar to the
inviscid time step, and is based on a “model” viscous problem, the heat diffusion

\(^3\)Other reasons are increased accuracy and reduced artificial dissipation.
equation [22]. The von Neumann number (VNN) is a nondimensional relationship for the viscous time step and is given in one dimension as,

\[
VNN = \nu \frac{\Delta t}{(\Delta x)^2}.
\]  

Following the inviscid time step formulation, the corresponding viscous time step is given in Equation (2.46), where it is now related to a viscous eigenvalue.

\[
\Delta t_v = \frac{VNN}{\lambda_v}
\]

The viscous eigenvalues, \(\lambda_v\), are the sum of the spectral radii (i.e. maximum eigenvalue) of the viscous fluxes. Using the primitive variable set \(Q_p = (\rho, u, v, w, p)^T\) the nondimensional Navier-Stokes equations are rewritten in a curvilinear coordinate system as,

\[
\frac{\partial Q_p}{\partial t} = \left[ \begin{array}{c} A_\xi \frac{\partial Q_p}{\partial \xi} + A_\eta \frac{\partial Q_p}{\partial \eta} + A_\zeta \frac{\partial Q_p}{\partial \zeta} + B_{\xi,\xi} \frac{\partial^2 Q_p}{\partial \xi^2} + B_{\eta,\eta} \frac{\partial^2 Q_p}{\partial \eta^2} + B_{\zeta,\zeta} \frac{\partial^2 Q_p}{\partial \zeta^2} \\
+ (B_{\xi,\eta} + B_{\eta,\xi}) \frac{\partial^2 Q_p}{\partial \xi \partial \eta} + (B_{\xi,\zeta} + B_{\zeta,\xi}) \frac{\partial^2 Q_p}{\partial \xi \partial \zeta} + (B_{\eta,\zeta} + B_{\zeta,\eta}) \frac{\partial^2 Q_p}{\partial \eta \partial \zeta} \end{array} \right]
\]

This expansion is based on a local linearization of the Navier-Stokes equations and is acceptable for stability analyses [101]. The inviscid flux Jacobians are given by the \(A\) matrix and the viscous flux Jacobians are given by the \(B\) matrix. The viscous eigenvalues are given below for arbitrary directions. The \(\alpha\) and \(\beta\) subscripts would represent \(\xi\), \(\eta\), or \(\zeta\).

\[
B_{\alpha,\alpha} : \quad \lambda_{(\alpha,\alpha)} = S_1 \max \left[ \frac{\gamma}{Pr} \frac{\gamma - 4}{3} \right]
\]

\[
B_{\alpha,\beta} : \quad \lambda_{(\alpha,\beta)} = S_1 \max \left[ \frac{\gamma}{Pr} \frac{\gamma - 7}{3} \frac{\gamma - 4}{3} \right]
\]
where,

\[ S_1 = \frac{M_\infty \mu}{Re_\infty \rho} \]  (2.50)

\[ \nabla \alpha \cdot \nabla \beta = \alpha_x \beta_x + \alpha_y \beta_y + \alpha_z \beta_z \]  (2.51)

\[ |\nabla \alpha| |\nabla \beta| = \sqrt{(\alpha_x^2 + \alpha_y^2 + \alpha_z^2) + (\beta_x^2 + \beta_y^2 + \beta_z^2)}. \]  (2.52)

Using the eigenvalues the time step is given for a three dimensional problem as,

\[ \Delta t_v = \frac{VNN}{\lambda_{(\xi,\xi)} + \lambda_{(\eta,\eta)} + \lambda_{(\zeta,\zeta)} + 2 \left( \lambda_{(\xi,\eta)} + \lambda_{(\xi,\zeta)} + \lambda_{(\eta,\zeta)} \right)} \]  (2.53)

For the two-dimensional time step, the \( \zeta \) terms are omitted.

Once the inviscid and viscous time steps are known they are combined to obtain the computational time step, given in Equation (2.54).

\[ \frac{1}{\Delta t} = \frac{1}{\Delta t_i} + \frac{1}{\Delta t_v} \]  (2.54)

Lastly, the time step can be written in terms of the eigenvalues and CFL and VNN numbers,

\[ \Delta t = \frac{CFL}{\lambda_i + \frac{CFL}{VNN} \lambda_v}. \]  (2.55)

With,

\[ \lambda_i = \lambda_\xi + \lambda_\eta + \lambda_\zeta \]  (2.56)

\[ \lambda_v = \lambda_{(\xi,\xi)} + \lambda_{(\eta,\eta)} + 2 \left( \lambda_{(\zeta,\xi)} + \lambda_{(\xi,\eta)} + \lambda_{(\eta,\xi)} + \lambda_{(\eta,\zeta)} \right). \]  (2.57)

The maximum stable CFL number for an explicit Runge-Kutta time integration is \( 2\sqrt{2} = 2.82 \). However, a value of 0.6 is suggested for use in CHOPA to ensure stability. The VNN is dependent upon the dimensions of the problem and the total VNN must sum to 0.5 for stability [103], so
\[
VNN = \begin{cases} 
0.5 & 1D \\
0.25 & 2D \\
1/6 & 3D 
\end{cases} \quad (2.58)
\]

### 2.3 Convergence Acceleration

Since the simulation of two-, and especially, three-dimensional cavity flows is computationally expensive due to the large number of grid points required to capture the strong gradients in the viscous boundary layers, several methods are proposed to accelerate the convergence. These methods are discussed below.

### 2.4 Preconditioning

To facilitate an efficient solution for this solver, preconditioning of the time derivative is employed. The preconditioning will dramatically accelerate the convergence rate of the code and also permit the solution of some problems that can be numerically unstable (which is a real issue for low Mach number flows). Preconditioning of the discretized equations can eliminate these difficulties by equalizing the acoustic wave and turbulence convection speeds through altering the governing differential equations. However, this destroys the time accuracy of the solution, which is unacceptable for highly unsteady flows such as those found in cavities. This drawback can be circumvented by using a dual time-stepping method to reintroduce time accuracy (See Section 2.2.2.1). Many different preconditioning methods have been examined and some will be highlighted below. This thesis will provide a background for preconditioning methods but detailed reviews of Euler and Navier-Stokes preconditioners, including stability analyses and solution methods, are given by Turkel \textit{et al.} \cite{104–107}, Venkateswaran and Merkle \cite{108,109}, Zaccanti and Cinnella \cite{110} and others \cite{111–113}.

For low Mach number flows the numerical solution of the Navier-Stokes equations becomes extremely difficult because the system of equations is very “stiff”. The stiffness in the system is due to the disparity between the acoustic wave speed \((u + c)\) and the turbulence convection speed \((u)\). The condition number is defined as the ratio of the maximum to minimum eigenvalues of the system, \((u + c)/u\),
and is a measure of the disparity. The greater the condition number the stiffer the system. Both explicit and implicit time integration schemes have difficulty solving stiff systems. When the condition number is close to one, the acoustic and convective speeds are similar and convergence is rapid. Occasionally, at very low flow speeds this slow convergence rate can even result in a numerically unstable problem [114]. For an optimal preconditioner, the new system results in a condition number as close to one as possible for all Mach numbers [115]. It should be noted that for turbulent flows preconditioning is not required for the turbulence models [116]. Other researchers have investigated the effect of preconditioning on turbulence models including Tong and Katz [117] and Djeddi et al. [118].

Since there is a large disparity in the acoustic wave and turbulence convection speeds in low speed flow, a time-step that would be appropriate to solve one wave would not be appropriate for the other, regardless of the method of time integration. An explicit scheme has a low operation count per iteration, but when used to solve the numerical system the time step is inversely proportional to the largest eigenvalue of the system, $u + c$. Thus, using the small $\Delta t$ from $u + c$, a tremendous number of time steps are required to propagate particles moving at the turbulence convection speed through the domain. Alternatively, if an implicit scheme is used to solve the numerical system, problems also occur with convergence. A fully implicit algorithm (where the full matrices are inverted) is still not feasible at this time due to the large size of matrices in conventional computational fluid dynamics (CFD) problems [119]. Instead, most implicit algorithms are based on approximately factored methods (see [22]) that have an optimum CFL number between one and ten. However, the optimum CFL number can only be maintained for one wave speed ($u$ or $u + c$) resulting in slow convergence for the other wave speed. Also, with an approximate factorization method, the inherent error due to factorization can drastically reduce the convergence rate [120]. Therefore an implicit solution method also results in low convergence rates for low Mach number problems.

The remedy for this convergence problem is to precondition the governing equations. There are two general types of preconditioners that are commonly employed: time derivative (also known as eigenvalue scaling) and asymptotic [121]. There are other methods that can be used to derive preconditioners [115, 122–125], but these are not discussed here. In the first approach, eigenvalue scaling, the range of wave
speeds is minimized by introducing a matrix that premultiplies the time derivatives of the equations, equalizing the acoustic and turbulence convective speeds. A subset of time derivative preconditioners are those for incompressible flow, which were the starting point for all types of preconditioning. The second approach uses a perturbation expansion method to derive compressible equations that are asymptotically valid in the low Mach number limit [126]. The asymptotic method develops new equations of motion based on an order of magnitude comparison of terms after replacing the acoustic waves with pseudo-acoustic waves that have the same speed as the convective waves.

It is interesting to note that, in general, the transpose of the preconditioner matrix, $\Gamma$, is also a valid preconditioner [107]. Since the determinant of a matrix is equal to the determinant of its transpose, the eigenvalues of both matrices will be the same, although the eigenvectors are different. So while they are both valid preconditioners, the orthogonality of the eigenvectors is an important property of a preconditioner. Near stagnation points some preconditioners artificially generate large amounts of vorticity that can cause instability [127]. Unless mentioned otherwise all preconditioners are given for $\Gamma$ that acts on the time derivative. Accordingly, $\Gamma^{-1}$ is used on the residual.

Even though the preconditioner used in this work is for the Navier-Stokes equations, the Euler equations can provide excellent insight into the preconditioning effects on a system and are almost always the basis for Navier-Stokes preconditioners. More detailed analyses on Euler preconditioners can be found in several references [110, 115, 127–129].

The next several sections will describe a large variety of preconditioners. These preconditioners vary in how they are applied to the governing equations, from operating on the time derivative to asymptotic expansions about the mean flow. The preconditioners are also broken out based on inviscid and viscous flows. In addition to the preconditioners the boundary conditions and artificial dissipation methods specific to preconditioned flows are briefly discussed.

2.4.1 Time Derivative Preconditioner

The earliest preconditioner is found in the artificial compressibility method for incompressible flow. In the incompressible equations there is no link between the
density (or pressure if non-conservative variables are used) and velocity in the time domain. So, an artificial time derivative of density is added to the continuity equation. The equations are then hyperbolic and can be solved with a variety of time marching techniques that have been developed for compressible flows. At convergence, the artificial time derivative is zero and the incompressible equations are recovered. Based on the excellent results of this work, preconditioning matrices were developed for the steady compressible equations, and eventually for the unsteady compressible equations.

2.4.1.1 Incompressible Preconditioner

The incompressible Navier-Stokes equations can be written in Cartesian tensor notation as,

\[
\frac{\partial u_i}{\partial x_i} = 0 \quad (2.59)
\]

\[
\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \quad (2.60)
\]

where \( t \) is time, \( x_i \) is the Cartesian coordinate, \( u_i \) are the velocity components, \( p \) is the pressure, and \( \tau_{ij} \) is the viscous stress tensor [83]. Note that Equations (2.59) and (2.60) completely describe the flow field; the energy equation is decoupled from the equations. This is an advantage in solving the incompressible as opposed to the compressible equations. Since the number of coupled equations is reduced by one, the computation time can often be reduced.

The seminal work in artificial compressibility was conducted by Chorin [130] who added a time derivative term to the incompressible continuity equation,

\[
\frac{1}{\beta} \frac{\partial p}{\partial \tau} + \frac{\partial u_i}{\partial x_i} = 0 \quad (2.61)
\]

where \( \tau \) is a fictitious time and \( 1/\beta \) is the artificial compressibility factor selected to achieve the maximum convergence rate. This term changes the incompressible Navier-Stokes equations from a mixed elliptic-parabolic set of equations to a mixed hyperbolic-parabolic set, where time-marching approaches can be used. Since the artificial compressibility method does not require a divergence free velocity field (as with the pressure correction method) the time integration method converges
more quickly. The artificial density, $\tilde{\rho}$, is related to the pressure term through an artificial equation of state, given by,

$$ p = \frac{\tilde{\rho}}{\beta} \quad (2.62) $$

The addition of the artificial compressibility term adds an artificial pressure wave and $\beta$ is often related to an artificial speed of sound,

$$ \tilde{c} = \sqrt{\beta} \quad (2.63) $$

where $\tilde{c}$ is the artificial speed of sound. The corresponding artificial Mach number is $\tilde{M} = u/\tilde{c}$.

Two parameters can be altered to accelerate convergence: the artificial time-step and the value of $\beta$. Since the time term, $\tau$, is a fictitious time the time-step can be of arbitrary size, which can be as large as possible (within stability constraints) to expedite the convergence to a steady state [131]. This is especially beneficial if an implicit method is chosen, where the $\Delta\tau$ allowed by stability constraints is theoretically infinite but is usually limited due to approximate factorization errors.

For a hyperbolic system of equations, the eigenvalues of $A$, see Equation (2.64), give the propagation information of the system [132].

$$ \frac{\partial Q}{\partial t} + A \frac{\partial Q}{\partial x} = 0 \quad (2.64) $$

From this, for the incompressible Navier-Stokes equations the eigenvalues of the standard and preconditioned equations are, respectively,

$$ \lambda = \begin{bmatrix} u - c \\ u \\ u \\ u \\ u + c \end{bmatrix} \quad (2.65) $$
\[
\lambda = \begin{bmatrix}
    u - \sqrt{u^2 + \beta} \\
    u \\
    u \\
    u \\
    u + \sqrt{u^2 + \beta}
\end{bmatrix}
\]

(2.66)

Again indicative that \( \beta \) is akin to the square of the speed of sound. The preconditioned eigenvalues are in slight contrast to the artificial speed of sound, in that they imply that the sound speed is \( \sqrt{u^2 + \beta} \), instead of \( \sqrt{\beta} \). However, remember that for low Mach number flow, \( u^2 \ll \beta \) and this approximation holds. If the speed of sound is much greater than the turbulence convective speed, as is the case for low Mach number flows, then this suggests that the magnitude of \( \beta \) should be close to the convective velocities \( (\beta^2 \approx (u^2 + v^2 + w^2)) \) to remove the stiffness in the system and enhance the rate of convergence [83].

Both explicit and implicit methods have been used to solve the Navier-Stokes equations with artificial compressibility. Kwak et al. [133], Zaccanti and Cinnella [127], Choi and Merkle [120], Rogers et al. [134], and Soh and Goodrich [131] discuss implicit solutions while explicit results are given by Merkle and Tsai [135]. More studies have been conducted for implicit methods, as artificial compressibility allows for a time-marching scheme to be used and, for steady state problems, a very large \( \Delta t \) can be used to arrive quickly at a solution. A more detailed review of incompressible preconditioners is given in references [104–108].

### 2.4.1.2 Compressible Preconditioner

The work of Turkel, collaborating with numerous authors expanded upon Chorin’s work to include the compressible equations. The majority of the work in time derivative preconditioners has been performed by three groups: Turkel and co-workers [105, 107, 136, 137], van Leer and co-workers [129, 138], and Merkle and co-workers [103, 139]. Other notable researchers include Briley et al. [140], Lee [141], Fiedler and Ashcroft [142] who extend the work of Turkel, and Djeddi et al. [118]
who explicitly include a turbulence model in the preconditioner for improved convergence.

To simplify the analysis, the two-dimensional Euler equations will be examined, but this can be extended to the Navier-Stokes equations. In general terms, the preconditioned Euler equation for compressible flow can be written in non-conservative form as [143],

$$\Gamma \frac{\partial \mathbf{Q}}{\partial t} + A \frac{\partial \mathbf{Q}}{\partial x} + B \frac{\partial \mathbf{Q}}{\partial y} = 0 \quad (2.67)$$

where $\mathbf{Q}$ is the solution vector, $A$ and $B$ are the flux Jacobians, and the preconditioner is $\Gamma$. The definition of $\Gamma$ takes many different forms but all involve a scaling of the equations so the eigenvalues of the system of equations are equalized. The matrices range from a direct application of Chorin’s artificial compressibility, where only the time derivative of the continuity equation is modified, to preconditioners that modify the time derivatives for each equation.

The majority of the preconditioners relate the preconditioning parameter to the local flow velocity, which can cause problems in stagnation regions. As the velocity approaches zero the eigenvectors of the systems become parallel, which can cause numerical instability [128]. To alleviate these problems a limiter is often employed for the preconditioning parameter, to prevent it from going below a value where numerical instability might occur.

The next logical extension to preconditioners was to the compressible flow equations. The addition of viscous terms within the Navier-Stokes equations creates additional complexity for the preconditioning. Now a stiff system of equations results for low cell Reynolds numbers, $Re_c = \lambda \Delta x / \nu$, as well as low Mach numbers. The use of the Navier-Stokes equations also led to an inviscid time-step based on the CFL number and a viscous time-step based on the VNN number, that must be balanced for efficient convergence. Lastly, the strong gradients that occur in viscous flows near solid boundaries necessitate the use of highly clustered grids where the aspect ratio of the grid cells near the boundary can be very large. In these situations the acoustic and convective wave speeds no longer need to be equalized, but rather the cell crossing times of the waves should be considered. These additional constraints often require a dependence on Reynolds number and aspect ratio for Navier-Stokes preconditioners in addition to the flow velocity dependence.
Thus, the viscous flux terms in the Navier-Stokes equations greatly complicate the analysis of preconditioners.

### 2.4.2 Asymptotic Expansion Preconditioner

The second type of preconditioner uses an asymptotic expansion method to provide accurate solutions to the Euler and Navier-Stokes equations for time-marching algorithms. The preconditioner provides scaling factors to ensure that the terms added do not swamp the derivatives that describe the physical processes. The asymptotic expansion method has been championed by Merkle [109,144] and can handle a wide range of Mach number, fluid media, heat and mass sources, and highly stretched grids. Deng et al. have used it on unstructured, overset grids for the analysis of propellers [145]. Balasubramanian and Chen have also developed an asymptotic preconditioner for general turbomachinery flows [146].

The asymptotic preconditioner is primarily used for low Mach number flow applications, but different parameters can enable it to work at a wide range of speeds. This allows for the solution of the flow around airfoils at a high angle of attack, where there are often large stagnation regions. The main advantages of this method is that it can be used for an arbitrarily small Mach number and the extension to the Navier-Stokes equations is sometimes easier than time derivative preconditioning.

Since this preconditioner is primarily used for viscous problems, the discussion for this preconditioner will be for the Navier-Stokes equations, but the Euler equations can be considered similarly. For simplicity the two-dimensional equations will be discussed, but extension to three-dimensions is straightforward. The two-dimensional Navier-Stokes equations can be written in conservative form using a viscous variable set, $Q_v = (p,u,v,w,T)^T$, as,

$$\frac{\partial Q}{\partial Q_v} \frac{\partial Q_v}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} = L Q_v$$

where $E$ and $F$ are the flux vectors, $L$ is an operator that provides the viscous terms, and the physical Jacobian of the time derivative is,
\[
\frac{\partial \mathbf{Q}}{\partial \mathbf{Q}_v} = \begin{bmatrix}
\rho_p & 0 & 0 & \rho_T \\
\rho_p u & \rho & 0 & u \rho_T \\
\rho_p v & 0 & \rho & v \rho_T \\
\rho_p h_0 - (1 - \rho h_p) & \rho u & \rho v & \rho_T h_0 + \rho h_T
\end{bmatrix}
\] (2.69)

where \( T \) is temperature, \( \rho_p = (\partial \rho / \partial p)_T \), \( \rho_T = (\partial \rho / \partial T)_p \), \( h_T = (\partial h / \partial T)_p \), \( h_p = (\partial h / \partial p)_T \), \( h \) is enthalpy, and \( h_0 \) is the stagnation enthalpy. The subscript of the derivative refers to a variable to be held constant for the derivative. The physical time derivatives in the Jacobian are replaced with artificial values that are scaled based on a perturbation expansion. The primitive variables are expanded about a small parameter, \( \varepsilon = \rho_r u_r^2 / p_r \), which for a perfect gas is \( \varepsilon = \gamma M_r^2 \). In this nomenclature the subscript \( r \) signifies a reference quantity and \( \gamma \) is the ratio of specific heats. As an example the pressure is expanded as,

\[
p = p_0 + \varepsilon p_1 + \varepsilon^2 p_2 + ... \quad (2.70)
\]

Once all variables are expanded like powers are collected. When this is done, it shows that except for the pressure term, which has a first order term, only zeroth order terms remain. The artificial variables are then selected to balance the dominant physical processes at low speeds:

A. **Inviscid.** Inviscid flow is convection dominated. Balance the convective terms and pressure gradient.

B. **Viscous.** Low cell Reynolds number is diffusion dominated. Balance the diffusive terms and pressure gradient.

C. **Unsteady.** Unsteady regions where the physical time derivatives control. Balance the physical time derivative and pressure gradient.

The exact form for the preconditioner that automatically switches between these regimes is found in [109]. This preconditioner is also derived for an arbitrary equation of state, which allows easy extension to any fluid medium.
2.4.3 Inviscid Preconditioner

Whether one is examining the Euler or Navier-Stokes equations, the difficulty in solving for low speed flows exists in both. As in incompressible preconditioning, the time derivative is multiplied by a factor to equalize the acoustic and turbulence convective speeds. Except, for the compressible equations, it is a matrix and not just an artificial compressibility constant affecting the continuity equation. Thus for compressible preconditioning all of the equations are modified.

The Euler equations in three-dimensional Cartesian coordinates are given in their standard vector form as,

\[
\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = 0
\]  

(2.71)

where sources and heat fluxes have been neglected [83]. If a flux splitting scheme is used (e.g. upwind spatial discretization) then the preconditioned Euler equations can be written in vector form as,

\[
\Gamma \frac{\partial Q}{\partial t} + A \frac{\partial Q}{\partial x} + B \frac{\partial Q}{\partial y} + C \frac{\partial Q}{\partial z} = 0
\]  

(2.72)

where \( A \), \( B \), and \( C \) are flux Jacobian matrices [22].

As mentioned before, three research groups have been responsible for the more popular inviscid preconditioners. The salient features from each group’s inviscid preconditioners are highlighted below. Additional information on each of these, as well as the detailed form of the preconditioner, can be found in Appendix D.

2.4.3.1 Turkel Preconditioner

One of the better known inviscid preconditioners was developed by Turkel and is based on artificial compressibility. A two parameter preconditioner, deemed Turkel87 [106,110,147], was initially derived for curvilinear coordinate systems using one of several different solution vectors. Since the Turkel87 preconditioner is in stream-aligned coordinates it is theoretically independent of flow angle and eliminates the numerical instability encountered in stagnation regions [127]. This is not realized in practice since most numerical simulations can not handle stream-aligned coordinates and flow angle dependence is reintroduced in the transformation. This has been used successfully with a flux vector splitting scheme by Puoti [148] and
Rong and Wei [149]. Tyliszczak and Deconinck [150] have used this preconditioner to solve several benchmark cases, including 2D and 3D driven cavities. Fiedler and DiMare added a Turkel preconditioner to the German Aerospace Center’s (DLR) TRACE code [151] and verified it for a wide range of applications.

To help alleviate this instability in stagnation regions, the Modified Turkel preconditioner [152] was developed. This preconditioner is one of the best for low Mach number, inviscid flows. Turkel and co-workers have also developed another preconditioner based on Turkel87, called Turkel93 [107, 153]. In a Cartesian coordinate system the preconditioner is,

\[
\Gamma_{T93} = \begin{bmatrix}
\frac{c^2}{\beta^2} & 0 & 0 & 0 & \delta \\
\frac{\alpha u}{\rho \beta^2} & 1 & 0 & 0 & 0 \\
\frac{\alpha v}{\rho \beta^2} & 0 & 1 & 0 & 0 \\
\frac{\alpha w}{\rho \beta^2} & 0 & 0 & 1 & 0 \\
0 & 0 & 0 & 0 & 1
\end{bmatrix}.
\]  

This is essentially the same preconditioner as developed by Choi and Merkle (CMv), discussed later, except that different values are used for the \(\delta\) preconditioning parameter. In the Choi-Merkle inviscid preconditioner (CMi) described later, some elements have been discovered that do not improve the convergence rate. These are the (1, 5) and (5, 1) elements in the three-dimensional preconditioning matrix. When these are removed only one term remains and it is very similar to Chorin’s artificial compressibility approach and is termed a Chorin/Turkel preconditioner since it was originally proposed by Turkel [152]. This is a robust preconditioner that does not suffer from any symmetry constraints due to only having one term.

For Turkel’s preconditioners, and for most others, the \(\beta\) value appears in three places: (1) the preconditioner matrix, (2) the calculation of the time-step and, (3) the artificial dissipation [143]. Each of these benefit by a judicious choice of \(\beta\). From numerical experiments, the preconditioner matrix based on the local velocity, with an appropriate limiter, is most effective.

2.4.3.2 van Leer Preconditioner

The thrust of van Leer and his coworkers has been to reduce the condition number using a local preconditioner. A major difficulty with the van Leer preconditioners
is that they are based in stream-aligned variables, which is acceptable only for simple flows. Zaccanti and Cinnella [110] have shown that it is quicker to perform the transformation analytically rather than within the numerical code. The transformation matrices to convert a streamwise preconditioner to a Cartesian coordinate system are found in Reference [114].

Darmofal and van Leer [115] developed a preconditioner in stream-aligned variables that has the lowest condition number and is called Optimal, see Appendix D. This preconditioner uses one preconditioning parameter and two arbitrary constants ($s$ and $\theta$) to develop other preconditioners that are a subset of the Optimal family. Based on this, van Leer et al. [129] developed the first preconditioner that is valid for all Mach numbers, VLR91 [154]. The VLR91 preconditioner is in the optimal family with $1/s = M\beta$ and $\theta = \pi$ and is presented in Cartesian coordinates,

$$
\Gamma_{VLR91}^{-1} = \begin{pmatrix}
\frac{M^2}{\beta} & -\frac{M}{\beta} \cos \theta & -\frac{M}{\beta} \sin \theta & 0 \\
-\frac{M}{\beta} \cos \theta & \left(\frac{1}{\beta} + 1\right) \cos^2 \theta + \beta \sin^2 \theta & \left(\frac{1}{\beta} + 1 - \beta\right) \sin \theta \cos \theta & 0 \\
-\frac{M}{\beta} \sin \theta & \left(\frac{1}{\beta} + 1 - \beta\right) \sin \theta \cos \theta & \left(\frac{1}{\beta} + 1\right) \sin^2 \theta + \beta \cos^2 \theta & 0 \\
0 & 0 & 0 & 1
\end{pmatrix}
$$

Where the parameter for optimization is $\beta = \sqrt{|1 - M^2|}$.

For VLR91 there is a slight decrease in convergence rate between two- and three-dimensional solutions. This is due to the introduction of shear waves that can rotate the flow, which can not be decoupled from the acoustic and turbulence convective speeds [129]. The determinant of VLR91 is $\tau^2/\beta^2 M^2$, showing that it is singular at $M = 0$, requiring a limiter to be used [127]. As expected, the VLR91 preconditioner has a flow angle dependence in it when transformed to Cartesian coordinate systems. This is responsible for problems in stagnation regions [127].

To enhance preconditioning an attempt at improving the eigenvector structure by enforcing a stream-wise orthogonality in the preconditioner was made. van Leer et al. [138, 155] developed a streamwise aligned preconditioner that has orthogonal eigenvectors in the streamwise direction called VL96. This preconditioner smoothly connects to VLR91 as the Mach number approaches one and is a member of the Optimal family. The VLR91 and Turkel preconditioners have been shown to be from the same family with the VL96 preconditioner related to the Turkel87 matrix.
for \( f = 0 \) \([138]\).

### 2.4.3.3 Choi-Merkle Preconditioner

The last significant inviscid preconditioner is by Merkle and his co-workers. The computational codes developed by this group focus on implicit time integration methods, so a primary concern is with the reduction of the approximate factorization errors using preconditioning. The preconditioner, CMi, only affects the energy equation and the preconditioner parameter is implicit through the total flow velocity used in the preconditioner \([156–158]\),

\[
\Gamma_{CMi} = \begin{bmatrix}
1 & 0 & 0 & 0 & 0 \\
0 & 1 & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 0 \\
0 & 0 & 0 & 1 & 0 \\
\frac{q^2}{2} \left( \frac{1}{M^2} - 1 \right) & u \left( 1 - \frac{1}{M^2} \right) & v \left( 1 - \frac{1}{M^2} \right) & w \left( 1 - \frac{1}{M^2} \right) & \frac{1}{M^2}
\end{bmatrix}
\]

(2.74)

This was written for a Cartesian coordinate system and is valid for a large variety of inviscid flows. It was also used as a basis for a later preconditioner, CMv, that can be used with either inviscid or viscous flows \([157]\). This preconditioner has been enhanced for a finite volume discretization method by Colin et al. \([159]\).

### 2.4.3.4 Jacobi Preconditioner

The Jacobi preconditioner is a matrix that arises naturally when upwind discretizations are performed on either the Euler or Navier-Stokes equations. The Jacobi preconditioner is different than the other preconditioners examined in that it does not reduce the condition number, but rather clusters the eigenvalues of the high frequency residual modes away from the origin. So, it is not specifically for low speed flows \([122, 160, 161]\). It is mentioned because it can be used in conjunction with an inviscid preconditioner to form a simple viscous preconditioner. After the preconditioner is applied to the system of equations one could choose the coefficients of a Runge-Kutta time integration method such that they effectively damp the high frequency waves.

This preconditioner was initially designed for multigrid methods since the high frequency errors must be removed from one grid before transferring the residuals to
a coarser grid [143, 162]. The high frequency components that can not be resolved on the next coarser grid must be damped before transferring the residual. Since a Jacobian preconditioner is effective in damping the high-high modes\(^4\) it must be combined with a semi-coarsening multigrid method for the high-low modes. More information on multigrid methods is given in Reference [163].

The Jacobi preconditioner works by clustering the eigenvalues in a region where they can be easily damped with an appropriate selection of time integration coefficients and is fairly robust [155]. When the eigenvalues are not clustered near the origin of the complex plane it is much easier to damp them. A disadvantage with Jacobi preconditioners is that a matrix must be inverted at each point, causing a significant increase in computational time [105]. Also, they do not increase the convergence rate for low Mach number flows and can create instabilities or are inaccurate. Finally, they can not handle high aspect ratio grids effectively. For these reasons a Jacobi preconditioner is not pursued in the present thesis.

\subsection*{2.4.4 Navier-Stokes Preconditioner}

As previously mentioned, the inclusion of viscous terms requires a more complex preconditioner. In the inviscid limit, the Navier-Stokes equations have a stiffness dependence on Mach number, like the Euler equations. As such, the preconditioning of the Navier-Stokes equations should be the same as the Euler preconditioner outside boundary layers. However, inside the boundary layer the viscous effects modify the eigenvalues, making them dependent on Reynolds number. When the cell Reynolds number is low, the imaginary part of the eigenvalue increases and a damping effect is produced. For cell Reynolds numbers less than one there is also a large disparity between the diffusive and convective time scales [164]. For acoustic dominated viscous flow, a mixture of Mach number and Reynolds number controls the stiffness. Additionally, at high frequencies, the imaginary parts of the waves can be all convective with no acoustic component, causing additional concerns for preconditioners [143]. Thus, any Navier-Stokes preconditioner must optimize the inviscid and viscous mode simultaneously by removing the Mach number and cell Reynolds number dependence from the solution.

\footnote{High-high refers to high frequency modes in the two coordinate directions. High-low and low-high modes are mixed modes where the coordinate directions have different frequency waves in the respective directions.}

47
There are two main methods that have been used to extend time derivative Euler preconditioners to the Navier-Stokes equations. In the first method, the viscous and heat conduction entries that arise naturally from viscous Jacobians are included along with the Euler preconditioner. This method is called viscous Jacobi preconditioning [165]. In the other method, the cell Reynolds number dependence is introduced into the Euler preconditioner according to a Navier-Stokes dispersion analysis [166].

The Navier-Stokes preconditioners are discussed assuming that they act upon the time derivative,

\[
\Gamma \frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = H
\]  

(2.75)

Where \( E, F, G \) include both the inviscid and viscous fluxes. This can also be written in nonconservative form as,

\[
\Gamma \frac{\partial Q}{\partial t} + A_i \frac{\partial Q}{\partial x} + B_i \frac{\partial Q}{\partial y} + C_i \frac{\partial Q}{\partial z} - D_v \frac{\partial Q}{\partial x} - E_v \frac{\partial Q}{\partial y} - F_v \frac{\partial Q}{\partial z} = H
\]  

(2.76)

where \( A_i, B_i, C_i \) are flux Jacobians and \( D_v, E_v, F_v \) are viscous Jacobians [22]. The \( H \) vector contains the source terms, which are taken as zero for all discussions. Since CHOPA uses central differencing for spatial derivatives, artificial dissipation is required for numerical stability. When the artificial dissipation, \( AD \), is included then the time derivative is equal to,

\[
\frac{\partial Q}{\partial t} = -\Gamma^{-1} \left[ \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} + H \right] + AD.
\]  

(2.77)

Optimizing both inviscid and viscous modes brings a considerable amount of complexity and requires both time steps to be examined and related to the preconditioning parameter. This is typically accomplished by equating the inviscid and viscous time steps to each other and solving for the preconditioning parameter. The preconditioning parameter is in the time step calculation since it appears in the inviscid eigenvalues. As discussed earlier, the inviscid time step is related to the CFL (Courant-Friedrich-Lewy) number while the viscous time step is related to the VNN (von Neumann number) constraint,
2.4.4.1 Time Derivative Preconditioner

2.4.4.1.1 Merkle Preconditioner  Merkle and Choi have developed a Navier-Stokes preconditioner that is based on a “viscous” set of primitive variables, \( Q_v = (p, u, v, w, T)^T \) called CMv [153, 157, 167]. These variables were selected due to time reversals that occurred for diffusive flows when using the CMi preconditioner in viscous flows and were inspired by incompressible flow variables as well as their work on perturbation expansions. In addition, Hauke and Hughes [168] have performed numerical experiments and concluded that this viscous variable set is the most robust one to use in numerical simulations. The choice of pressure as a variable is important because it aids in singling out the acoustic waves and makes the preconditioner dramatically simpler. The preconditioner modifies the continuity and energy equations while leaving the momentum equations unchanged.

The CMv preconditioner was expanded in 1993 to allow for a \( \delta \) parameter used for additional convergence control (\( \delta = 1 \) for the 1991 preconditioner version [167]). The selection of \( \delta = 0 \) does not change the convergence rate significantly. CMv is given in a Cartesian coordinate system as\(^5\),

\[
\Gamma_{CMv} = \begin{bmatrix}
\frac{1}{\beta M^2} & 0 & 0 & 0 \\
\frac{u}{\beta M^2} & \rho & 0 & 0 \\
\frac{v}{\beta M^2} & 0 & \rho & 0 \\
\frac{w}{\beta M^2} & 0 & 0 & \rho \\
\frac{e+p}{\rho \beta M^2} - \delta & \rho u & \rho v & \rho w & \frac{\gamma \rho R}{\gamma - 1}
\end{bmatrix}.
\]

This preconditioner is similar to the VM preconditioner (See Equation (2.96)) and is good for both explicit and implicit time integration methods, although Merkle prefers implicit methods. The CMv preconditioner can be linked to the Turkel93 preconditioner where \( \delta = 0 \) for Turkel93 and \( \delta = 1 \) for CMv [153]. The selection of \( \delta \) by Turkel simplifies the eigenvectors. Turkel also uses entropy instead of temperature in the solution variables.

For efficient selection of the time-step the CFL and VNN must both be controlled to enhance convergence. This led Buelow et al. [103, 169] to expand the CMv preconditioner to CMv-Buelow, that automatically accounts for inviscid and viscous regions, including high grid aspect ratios through an additional parameter,

\(^5\)N.B. For a perfect gas \( C_p = \frac{\gamma \rho R}{\gamma - 1} \) and \( \frac{e+p}{\rho} = h + \frac{1}{2} q^2 \)

49
The preconditioner is given in a Cartesian coordinate system as,

\[
\Gamma_{CMv-B} = \begin{bmatrix}
\frac{1}{\varepsilon c^2} & 0 & 0 & 0 \\
\frac{\rho}{\varepsilon c^2} & \rho & 0 & 0 \\
\frac{v}{\varepsilon c^2} & 0 & \rho & 0 \\
\frac{w}{\varepsilon c^2} & 0 & 0 & \rho \\
\frac{h + \frac{q^2}{\varepsilon c^2} - \delta}{\varepsilon c^2} - \delta & \rho u & \rho v & \rho w & \rho C_p
\end{bmatrix}
\]

(2.79)

2.4.4.1.2 Weiss/Smith Preconditioner

Weiss and Smith [90] have developed a preconditioner, WS, that does not assume an ideal gas in the derivation. This preconditioner continues to see significant use by researchers including for driven cavity flow by Li and Xiang [170]. This was derived in primitive variables based on pressure so that the acoustic wave speed could be controlled directly by an appropriate setting of the preconditioner. When viewed in symmetrizing variables this preconditioner is essentially the same as the Chorin/Turkel Euler preconditioner [121]. In a Cartesian coordinate system the preconditioner is given as,

\[
\Gamma_{WS} = \begin{bmatrix}
\beta & 0 & 0 & 0 & \rho_T \\
\beta u & \rho & 0 & 0 & \rho_T u \\
\beta v & 0 & \rho & 0 & \rho_T v \\
\beta w & 0 & 0 & \rho & \rho_T w \\
\beta h_0 - 1 & \rho u & \rho v & \rho w & \rho_T h_0 + \rho C_p
\end{bmatrix}
\]

(2.80)

where \( \rho_T = \frac{\partial \rho}{\partial T} \bigg|_p \), \( \rho_p = \frac{\partial \rho}{\partial p} \bigg|_T \), and \( C_p \) is the specific heat at constant pressure. The preconditioning parameter is defined as,

\[
\beta = \left( \frac{1}{U_r} - \frac{\rho_T}{\rho C_p} \right)
\]

(2.81)

It should be noted that when presented in symmetric variables a link can be established between CMi, CMv, and WS preconditioners [115]. This “link” preconditioner is given by,
The three preconditioners are linked by the following parameters,

\[ CM_i : \delta_{cmi} = 1, \delta_{cmv} = 0 \]
\[ CM_v : \delta_{cmi} = 0, \delta_{cmv} = 1 \]
\[ WS : \delta_{cmi} = 0, \delta_{cmv} = 0. \]

### 2.4.4.1.3 Pletcher/Chen Preconditioner

Pletcher and Chen [171, 172] developed a viscous preconditioner that uses a coupled modified strongly implicit procedure (CMSIP) for the solution. They write the equations in two dimensions in a strong conservative form using the viscous variable set. This preconditioner removes the ill-conditioned Mach number term in the continuity equation by dividing through by \( \gamma M^2 \). In a Cartesian coordinate system the preconditioner is written as,

\[
\Gamma_{PC} = \begin{bmatrix}
\frac{1}{T} & 0 & 0 & -\frac{p}{RT^2} \\
\frac{u}{T} & \frac{p}{RT} & 0 & -\frac{pu}{RT^2} \\
\frac{v}{T} & 0 & \frac{p}{RT} & -\frac{pv}{RT^2} \\
\frac{1}{\gamma} + \frac{1}{2} \frac{q^2}{C_p T} & \frac{pu}{RC_p T} & \frac{pv}{RC_p T} & \frac{pq^2}{2C_p RT^2}
\end{bmatrix}
\]

(2.83)

The use of the pressure in the solution vector again helps to alleviate the problems with an almost constant density that occurs for low Mach number flows.
2.4.4.2 Asymptotic Preconditioner Section

Another type of viscous preconditioner that can be used is an asymptotic preconditioner. For this, special perturbed forms of the governing equations are derived that are usually only valid at low Mach numbers. However, it is possible to extend this to moderate subsonic and transonic flows [126]. In the derivation of the perturbed forms of the equations of motion certain terms are assumed negligible, while others are added so that the acoustic waves can be replaced by pseudo-acoustic waves to equalize all the eigenvalues [119]. The advantages of this preconditioner over the time derivative approach is that it can be used for an arbitrarily small Mach number and its extension to the Navier-Stokes equations is simplified. The work in this area has almost been exclusively performed by Merkle and his co-workers.

The approach was initially developed for the Euler equations by Guerra and Gustaffson [173] using an expansion in terms of the Mach number to symmetrize the matrices of the flux Jacobians. This was later improved upon by Merkle and Choi [174, 175] using a perturbation expansion about Mach number squared to equalize the disparity in acoustic and convective wave speeds. The Mach number squared approach is better suited to the analysis since the momentum flux scales with Mach number squared. The acoustic waves are removed through the perturbation expansion and pseudo-waves are developed by adding an artificial time derivative to the energy equation. This term was added to the energy equation since the perturbation expansion removed all time derivatives and the derivatives are required for a time-marching solution method.

2.4.4.2.1 Merkle 1988  This asymptotic preconditioner is termed M88 [119]. As in the case of the Euler preconditioner, the Navier-Stokes equations for low Mach number are modified by expanding the nondimensional equations in a power series about a small parameter, $\varepsilon = \rho_r u_r^2 / p_r$. This is the ratio of dynamic pressure to thermodynamic pressure, which is a small value for low speed flow [126]. For a perfect gas this reduces to $\varepsilon = \gamma M_r^2$, where $\gamma$ is the ratio of specific heats and $M_r$ is a reference Mach number.

Then, starting from the nondimensional Navier-Stokes equations expressed in a viscous solution set, $Q_v$, a Mach number expansion of variable is introduced. The like powers of $\varepsilon$ are collected and a new system of equations are developed.

$^6$The subscript $r$ denotes the reference quantity used in the nondimensionalization.
However, except for the pressure, only the zeroth order variables remain and only the momentum and energy equations are altered. Due to this, the zeroth subscript is omitted from all variables in the asymptotic preconditioner. An additional constraint is developed from the $1/\varepsilon$ term that premultiplies the pressure gradient in the momentum equation, implying that the zeroth order pressure (or thermodynamic pressure) must satisfy,

$$\nabla p_0 = 0.$$ \hfill (2.84)

Thus the thermodynamic pressure is assumed constant for steady state problems. For the zeroth order Navier-Stokes equations the variables are $(p_1/\beta, u, v, T)^T$, but there is no time derivative to update the $p_1$ term. Based on Chorin’s artificial compressibility a time derivative for $p_1$ is inserted into the continuity equation,

$$\frac{\rho}{p_0} \frac{\partial}{\partial t} \left( \frac{p_1}{\beta} \right) + \frac{\partial \rho u_i}{\partial x_i} = 0$$ \hfill (2.85)

where $\beta$ is a parameter used for scaling the wave speeds. The preconditioner can be written in a Cartesian coordinate system as,

$$\Gamma_{M88} = \begin{bmatrix} \frac{\rho}{p_0} & 0 & 0 & 0 \\ \frac{\rho u}{p_0} & \rho & 0 & 0 \\ \frac{\rho v}{p_0} & 0 & \rho & 0 \\ 1 & 0 & 0 & \rho \end{bmatrix}$$ \hfill (2.86)

This provides an algorithm that has convergence rates almost independent of the Mach number of the flow. For inviscid flows $\beta$ is selected to equalize the acoustic and turbulence convective speeds, while for viscous flows $\beta$ is used to equalize the inviscid and viscous time steps. This is accomplished with the following definition.

$$\beta = \max \left\{ 1, \frac{u^2}{4T} \left[ \left( \frac{2 \text{CFL}}{\text{VNN \text{Re}_{\Delta x}}} - 1 \right)^2 - 1 \right] \right\}$$ \hfill (2.87)

### 2.4.4.2.2 Merkle Asymptotic

The information gained in developing the M88 preconditioner was extended to the Masymp preconditioner for improved conver-
gence of many different types of flows [92, 108, 176–178]. This preconditioner can efficiently handle low Mach number inviscid and viscous flows through a selection of preconditioning parameters based on fluid dynamic theory. For its derivation the Navier-Stokes equations can be written with the viscous solution vector, \( Q_v \), and the standard flux vectors as,

\[
\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = L_v Q_v
\]  

(2.88)

where \( E, F, G \) are the Euler flux vectors and \( L_v \) is an operator on the viscous solution vector that results in the viscous flux vectors [176]. The physical Jacobian \( \frac{\partial Q}{\partial Q_v} \) is,

\[
\frac{\partial Q}{\partial Q_v} = \begin{bmatrix}
\rho_p & 0 & 0 & 0 & \rho_T \\
u \rho_p & \rho & 0 & 0 & u \rho_T \\
v \rho_p & 0 & \rho & 0 & v \rho_T \\
w \rho_p & 0 & 0 & \rho & w \rho_T \\
h_0 \rho_p - (1 - \rho h_p) & \rho u & \rho v & \rho w & h_0 \rho_T + \rho h_T
\end{bmatrix} \]  

(2.89)

where a general equation of state is assumed \( \rho = \rho(p, T) \). For the preconditioning, the physical variables in the Jacobian, \( \frac{\partial Q}{\partial Q_v} \), are replaced with artificial variables resulting in the Masymp preconditioner in a Cartesian coordinate system,

\[
\Gamma_{Masymp} = \begin{bmatrix}
\rho'_p & 0 & 0 & 0 & \rho'_T \\
u \rho'_p & \rho & 0 & 0 & u \rho'_T \\
v \rho'_p & 0 & \rho & 0 & v \rho'_T \\
w \rho'_p & 0 & 0 & \rho & w \rho'_T \\
h_0 \rho'_p - (1 - \rho h_p) & \rho u & \rho v & \rho w & h_0 \rho'_T + \rho h_T
\end{bmatrix} \]  

(2.90)

Here \( h_0 \) is the total enthalpy, \( k_T \) ranges from zero for a perfect gas to one for an arbitrary equation of state, and
\[ h_T = \frac{\partial h}{\partial T} \left|_p \right. \quad \rho_p = \frac{\partial \rho}{\partial p} \left|_T \right. \]
\[ h_p = \frac{\partial h}{\partial p} \left|_T \right. \quad \rho_T = \frac{\partial \rho}{\partial T} \left|_p \right. \] (2.91)

For the preconditioned case, a perturbation expansion with respect to \( \varepsilon \) is performed on \( Q_v \) to result in a zeroth order Navier-Stokes equation with a solution vector of \( Q_v' = (p_1, u_0, v_0, w_0, T)^T \). A byproduct of this expansion is that \( p_0 \) is again equal to a constant. The preconditioning parameters are defined by examining the eigenvalues of the preconditioned system. The unpreconditioned eigenvalues are obtained from the solution of the homogeneous equation [177],
\[ \frac{\partial Q_v}{\partial Q} A_v - \lambda I = 0 \] (2.92)
which results in the standard eigenvalues, \( \lambda = \{u, u, u, u \pm c\}^T \), with the speed of sound as,
\[ c^2 = \frac{\rho h_T}{\rho_T(1 - \rho h_p) + \rho \rho_p h_T} \] (2.93)

The continuity, momentum, and energy equations are scaled to ensure that neither the convective nor viscous terms overwhelm the temporal and spatial derivatives to establish well conditioned equations. Based on these analyses the preconditioning parameters are selected to equalize the acoustic and convective wave speeds. There are two sets of preconditioning parameters that have been derived for use in the Masymp preconditioner, with similar results.

### 2.4.4.3 Other Viscous Preconditioners

While there are other methods that have been developed for preconditioning, these have not received much attention, mostly due to the success of the time derivative and asymptotic methods. One method developed by Eriksson is based on an isentropic pressure-under-relaxation concept [179]. This method changes the rate of change of pressure to reduce the acoustic waves to the convective speed. Turkel has also developed a differential preconditioner that includes partial derivatives [104]. The derivatives are used to account for grid aspect ratios far from one
and for when the flux Jacobians do not commute. If the flux Jacobians do not
commute, the wave speeds can not truly be equalized since the acoustic wave and
turbulence convective waves can not be exactly separated. Other preconditioners
are described in Appendix D.

2.4.4.3.1 Viscous Jacobi Preconditioner A method used to develop a pre-
conditioner for the Navier-Stokes equations is to add the viscous and heat conduc-
tion entries from the viscous Jacobian to an existing Euler preconditioner [165,180].
This results in a preconditioner that varies from an Euler preconditioner to a vis-
cous dominated preconditioner depending on the cell Reynolds number. This pre-
conditioner is well suited to handling partial differential equation-based turbulence
modeling since it can deal with large aspect ratios and source terms. However, this
method is considered point-implicit due to the analytical difficulty in inverting the
preconditioner and it can be time consuming.

Turkel Preconditioner Turkel et al. [136] used a combined implicit-explicit
time integration method with the Turkel93 preconditioner to analyze low Mach
number flows. The Euler preconditioner was simply extended to the viscous flow
regime by including the viscous Jacobi terms in the Navier-Stokes preconditioner.
Good results were obtained with this method. This is given in three dimensions
as,

$$\Gamma_{TVJ} = \Gamma_{T93} + 2 \left( \frac{D_v}{\Delta x^2} + \frac{E_v}{\Delta y^2} + \frac{F_v}{\Delta z^2} \right)$$

(2.94)

van Leer Preconditioner van Leer and his co-workers [138,180] have also de-
veloped a viscous Jacobi preconditioner that bases the inviscid portion on a modifica-
tion of the VLR91 preconditioner that to account for the high aspect ratio of cells
in a viscous boundary layer. The aspect ratio dependence is based on a projection
of the velocity on each face. The preconditioners are called V-VJ97 and V-VJ99
and are very similar (see Appendix D). In a stream-aligned, two-dimensional co-
ordinate system V-VJ97 is given as,
\[ \Gamma_{V-VJ97} = \Gamma_{Eu} + \frac{1}{\alpha} \left( \frac{2D_v}{\Delta x^2} + \frac{2E_v}{\Delta y^2} - \Delta t \frac{\partial H}{\partial Q} \right) \] (2.95)

where \( \Gamma_{Eu} \) is the VLR91 preconditioner. The last term in the preconditioner, \( \Delta t \frac{\partial H}{\partial Q} \), can be used to account for the source terms in turbulence modeling [138].

From numerical experiments Lee [147] has found that the VLR91 and Chorin types of Euler preconditioners work best with the viscous Jacobians. Lee has subsequently derived a viscous Jacobian preconditioner, based on the VLR91 Euler preconditioner, and incorporated it into V-VJ99.

**Dispersion Analysis Preconditioner**  Another method that extends an Euler preconditioner to the Navier-Stokes equations involves a Fourier analysis of the linearized equations to arrive at an analytic solution. This is difficult and has only been performed exactly for the one-dimensional case with simplifying assumptions [138, 166]. The object with the dispersion analysis is still to reduce the condition number to one.

Based on a dispersion analysis, Venkateswaran and Merkle [166] have extended one of their asymptotic preconditioners (Merkle Asymptotic). The Venkateswaran and Merkle preconditioner, VM, is given as,

\[
\Gamma_{VM} = \begin{bmatrix}
\frac{1}{\varepsilon c^2} & 0 & 0 & 0 & 0 \\
\frac{u}{\varepsilon c^2} & \rho & 0 & 0 & 0 \\
\frac{v}{\varepsilon c^2} & 0 & \rho & 0 & 0 \\
\frac{w}{\varepsilon c^2} & 0 & 0 & \rho & 0 \\
\frac{h + \frac{1}{\varepsilon c^2}}{\varepsilon c^2} - \delta & \rho u & \rho v & \rho w & \rho C_p
\end{bmatrix}
\] (2.96)

with the preconditioning parameter defined by,

\[
\varepsilon = \begin{cases}
M^2, & \text{Re}_{\Delta x} \gg 1 \\
M^2, & \text{Re}_{\Delta x} \ll 1, \frac{M^2}{\text{Re}_{\Delta x}} \ll 1 \\
\frac{1}{\text{Re}}, & \text{Re}_{\Delta x} \ll 1, \frac{M^2}{\text{Re}_{\Delta x}} \gg 1
\end{cases}
\] (2.97)

In a similar fashion Lee [147] developed a viscous preconditioner that uses a dis-
persion analysis with a viscous Jacobian, $\Gamma_{Lee-NS}$ and an extension to the VLR91 preconditioner. van Leer [138] has also used the dispersion analysis to develop a Chorin-type preconditioner that only has a (1, 1) element. This preconditioner is given as $\Gamma_{V_{LDA}}$ in Appendix D.

2.4.4.4 Preconditioner of Choice

The preconditioner that has been selected for implementation in the present thesis is Buelow’s extension to the Choi-Merkle preconditioner, CMv-B [103, 139]. They developed two preconditioners that give the same eigenvalues, but different eigenvectors. Just one is selected for implementation since different eigenvectors have different characteristics and any formulation requiring characteristic information (e.g. artificial dissipation or boundary conditions) will be dependent on the preconditioner used. Neither has been found to have an advantage over the other in flow simulations. They can be derived for an arbitrary fluid but are shown here assuming a perfect gas. This time derivative preconditioner is selected as it is applicable over a wider range of Mach numbers than asymptotic preconditioners.

The preconditioner, designed to act on the time derivative and using the viscous variable set, is given in dimensional form as,

$$
\Gamma_{CMv-B} = \begin{bmatrix}
\frac{1}{\varepsilon RT} & 0 & 0 & 0 & -\frac{\rho}{T} \\
\frac{u}{\varepsilon RT} & \rho & 0 & 0 & -\frac{\rho u}{T} \\
\frac{v}{\varepsilon RT} & 0 & \rho & 0 & -\frac{\rho v}{T} \\
\frac{w}{\varepsilon RT} & 0 & 0 & \rho & -\frac{\rho w}{T} \\
\frac{\rho u + \rho E_t}{\rho \varepsilon RT} & -\frac{1}{\varepsilon} & \rho u & \rho v & \rho w & -\frac{\rho}{T} \left[ \frac{\rho u + \rho E_t}{\rho} - C_p T \right]
\end{bmatrix}.
$$

(2.98)

Where $C_p^7$, is the specific heat at constant pressure and $\varepsilon$ is the preconditioning parameter.

Now, for implementation in a computational aeroacoustics code the preconditioner must be rederived in nondimensional variables and for a generalized coordinate system. This has not been presented in detail in the open literature and is shown here. The resulting preconditioner is,

\[7\text{N.B. For a perfect gas } C_p = \frac{2\gamma R}{\gamma-1} \text{ and } \frac{E_t + p}{\rho} = h + \frac{1}{2}q^2.\]
Here $e$ is the internal energy per unit volume. The preconditioner is controlled through the preconditioning parameter, $\varepsilon$. A beneficial aspect of this preconditioner is that when $\varepsilon = 1$ the unpreconditioned Navier-Stokes are recovered. This is particularly helpful in debugging.

The Choi-Merkle preconditioners were originally developed using the solution variable $Q_v = (p, u, v, w, T)$, where $p$ is the thermodynamic pressure and $T$ is the temperature, so the Jacobian $\frac{\partial Q_v}{\partial Q_v}$ is implicit within the preconditioner. Thus, for ease of implementation the preconditioner is postmultiplied by a Jacobian, $\frac{\partial Q_v}{\partial Q_v}$, so that the preconditioner ultimately acts on the conservative variable set. The preconditioner then premultiplies the time derivative, resulting in,

$$
\Gamma_{CMv-B} = \begin{bmatrix}
\frac{1}{\varepsilon(e(\gamma-1)} & 0 & 0 & 0 & -\frac{\rho}{T} \\
\frac{u}{\varepsilon(e(\gamma-1))} & \rho & 0 & 0 & -\frac{\rho u}{T} \\
\frac{v}{\varepsilon(\gamma-1)} & 0 & \rho & 0 & -\frac{\rho v}{T} \\
\frac{w}{\varepsilon(\gamma-1)} & 0 & 0 & \rho & -\frac{\rho w}{T} \\
\frac{(p+\rho E_t)\gamma}{\rho e} & -\frac{1}{\varepsilon} & \rho u & \rho v & \rho w & -\frac{\rho}{T} \left[ \frac{p+\rho E_t}{\rho} - \frac{T}{\gamma-1} \right]
\end{bmatrix}
$$

(2.99)

Lastly, the preconditioner acts on the residual in a numerical code, requiring the inverse of the preconditioner presented here. For a generalized coordinate system, using conservative variables for the nondimensional Navier-Stokes equations and acting on the residual, is given as,

$$
\Gamma_{CMv-B} \frac{\partial Q_v}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = H \quad (2.100)
$$

$$
\Gamma_{CMv-B} \frac{\partial Q_v}{\partial Q_v} \frac{\partial Q_v}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = H \quad (2.101)
$$

$$
\Gamma \frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = H. \quad (2.102)
$$
\[
\Gamma^{-1} = \frac{1}{z_2} \begin{bmatrix}
z_3 - q^2 z_1 + 1 & uz_1 & vz_1 & wz_1 & -z_1 \\
-q^2 z_1 u & z_2 + u^2 z_1 & uwz_1 & -uz_1 \\
-q^2 z_1 v & uwz_1 & z_2 + v^2 z_1 & -vz_1 \\
-q^2 z_1 w & uwz_1 & vwz_1 & z_2 + w^2 z_1 & -wz_1 \\
-q^2 z_3 & uz_3 & vz_3 & wz_3 & -q^2 z_1 + 1
\end{bmatrix} \tag{2.103}
\]

with,

\[
q^2 = u^2 + v^2 + w^2 \tag{2.104}
\]

\[
z_1 = \frac{1}{\varepsilon e} (1 - \varepsilon) \tag{2.105}
\]

\[
z_3 = z_1 E_t \tag{2.106}
\]

\[
z_4 = z_3 - \frac{q^2}{2} z_1 \tag{2.107}
\]

\[
z_2 = 1 + z_4 \tag{2.108}
\]

### 2.4.4.5 Pressure/Density Decomposition

At very low Mach numbers it is difficult to calculate the density and pressure gradients accurately due to the extremely small variations in these variables, often causing machine round-off errors that swamp the calculation. This results in a residual that decreases to a nonconverged level and then plateaus, preventing convergence.

In addition to preventing convergence as the Mach number decreases the numerical accuracy of the solution also decreases [181]. For a Mach number lower than approximately 0.1 the degradation in solution accuracy becomes evident and for Mach numbers below \(10^{-3}\) is significant [132, 157]. It is also known that numerical accuracy is improved for low Mach number flows when a preconditioner is used. A numerical study conducted by Volpe [181] showed that for nonpreconditioned fluid dynamics codes the error is inversely proportional to \(\sqrt{M}\). In general, upwind schemes tend to have greater errors for low Mach number flows [182].

In support of this, Reed [132] has proved that the truncation error from the spatial discretization is higher for a nonpreconditioned system than for a precon-
ditioned system. If the truncation error is $TE$, then the difference between non-preconditioned and preconditioned errors is given by,

$$ T E_{np} = \frac{1}{M^2} T E_p $$

To correct this problem a constant value can be subtracted from the density and pressure. Choi and Merkle use a gauge pressure of $p = \bar{p} + p'$ where $\bar{p}$ is an arbitrary constant (e.g. freestream value) with the solution vector $Q'_v = (p', u, v, T)^T$. Turkel [104] also advocates the use of subtracting a constant pressure from the dynamic pressure. This makes the fluctuations more significant but, is only performed for isolated variables. The pressure term in the energy equation remains unchanged [157]. Those that are multiplied by other variables (e.g. $\rho u$ in the momentum equation) do not require this manipulation. Thus, the preconditioned Navier-Stokes equations are rewritten as,

$$ \Gamma \frac{\partial Q'_v}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{F}}{\partial y} + \frac{\partial \mathbf{G}}{\partial z} = \mathbf{H} $$

where

$$ Q'_v = \begin{bmatrix} \rho' \\ \rho u \\ \rho v \\ \rho w \\ \rho E_t \end{bmatrix}, \quad \mathbf{E} = \begin{bmatrix} \rho u \\ \rho u^2 + p' - [\tau_{xx}] \\ \rho w - [\tau_{xy}] \\ \rho w - [\tau_{xz}] \\ (\rho E_t + p) u - [u\tau_{xx} + v\tau_{xy} + w\tau_{xz} - q_x] \end{bmatrix}, $$

$$ \mathbf{F} = \begin{bmatrix} \rho v \\ \rho v u - [\tau_{xy}] \\ \rho v^2 + p' - [\tau_{yy}] \\ \rho v w - [\tau_{yz}] \\ (\rho E_t + p) v - [u\tau_{xy} + v\tau_{yy} + w\tau_{yz} - q_y] \end{bmatrix}, $$
\[
G = \begin{cases}
\rho w \\
\rho uw - [\tau_{xz}] \\
\rho vw - [\tau_{yz}] \\
\rho w^2 + p' - [\tau_{zz}] \\
(\rho E_t + p) w - [u\tau_{xz} + v\tau_{yz} + w\tau_{zz} - q_z]
\end{cases}
\] 

(2.111)

where \( \rho' \) is \( \rho - \rho_\infty \) and \( p' \) is \( p - p_\infty \). This is efficiently incorporated into a code by changing to the decomposed variables immediately before calculating the residual, then reverting back to the standard conservative variables after the residual is calculated.

2.4.4.6 Selection of Preconditioning Parameter, \( \varepsilon \)

The concept of preconditioning was initially developed for inviscid flows to equalize the wave speeds and increase the convergence rate of the solution. Unlike viscous problems, the solution of inviscid flows does not require high aspect ratio grid cells, which are required to accurately predict the high gradients in the boundary layer. When the preconditioner is applied to viscous flows, additional measures must be used to remove the Mach number and cell Reynolds number effects on the convergence rate.

For inviscid flows there are no viscous terms, so high aspect ratio grids and the preconditioning parameter is directly related to Mach number. However, efforts must be used to prevent singularities in the preconditioner matrix when the Mach number approaches zero in stagnation regions [110]. For these flow regimes a limiter suggested by Buelow [103] is applied to the preconditioning parameter. This is given in Equation (2.112) and the limiting value is set to \( 1 \times 10^{-5} \). This means that effective preconditioning will not occur in regions where \( M < 1 \times 10^{-5} \), but that is not expected to be a significant portion of most flow fields. Lee et al. [183] also provide a formulation that switches the local time stepping based on min-CFL/max-VNN methods but that is not used for this work.

\[
\varepsilon_{\text{inv}} = \begin{cases}
\varepsilon_{\text{lim}}^2 & M < \varepsilon_{\text{lim}} \\
M^2 & \varepsilon_{\text{lim}} < M < 1 \\
1 & M > 1
\end{cases}
\]

(2.112)
The simultaneous optimization of inviscid and viscous flows is accomplished by utilizing both types time steps and incorporating the Mach number effects. The inviscid time step is related to the CFL (Courant-Friedrich Lewy) number while the viscous time step is related to the VNN (von Neumann) number. The description below will be for an arbitrary coordinate direction. In the formulation, the variable \( L \) can be replaced with \( \xi \), \( \eta \), or \( \zeta \) for the specific direction of interest.

The inviscid time step calculation is based on the eigenvalues of the preconditioned Euler equations, the same as in Section 2.2.2.4 except it is for a single direction only. This is because \( \varepsilon \) cannot be explicitly solved for when multiple directions are considered. All of the viscous eigenvalues are kept since they are not dependent on the preconditioning parameter. It is assumed that the calculation is being performed on a uniform grid in computational space, so all grid spacing is equal to one (\( i.e. \Delta L = 1 \)). Thus,

\[
\Delta t_i = \frac{\text{CFL} \Delta L}{\max (\lambda_L)} = \frac{\text{CFL}}{\frac{1}{2} [U(1 + \varepsilon) + C']}
\]

\[
C' = \sqrt{U^2 (1 - \varepsilon)^2 + 4\varepsilon c^2 \left(L_x^2 + L_y^2 + L_z^2\right)}.
\]

(2.113)

(2.114)

and,

\[
\Delta t_v = \frac{\text{VNN}}{\lambda_{(\xi,\xi)} + \lambda_{(\eta,\eta)} + \lambda_{(\zeta,\zeta)} + 2 \left(\lambda_{(\xi,\eta)} + \lambda_{(\xi,\zeta)} + \lambda_{(\eta,\zeta)}\right)}
\]

(2.115)

For each coordinate direction, the inviscid and viscous time steps are set equal to each other and \( \varepsilon \) is found. The directional \( \varepsilon \) is given by,

\[
\varepsilon_{\text{vis}}^L = \frac{\delta (\delta - U)}{U (\delta - U) + c^2 |\nabla L|^2}
\]

(2.116)

with,

\[
\delta = \frac{\text{CFL}}{\text{VNN}} \left[\lambda_{(\xi,\xi)} + \lambda_{(\eta,\eta)} + \lambda_{(\zeta,\zeta)} + 2 \left(\lambda_{(\xi,\eta)} + \lambda_{(\xi,\zeta)} + \lambda_{(\eta,\zeta)}\right)\right]
\]

(2.117)

\[
U = L_x u + L_y v + L_z w
\]

(2.118)

The final viscous parameter is selected by using the maximum value from all available directions,
\[ \varepsilon_{vis} = \max \left[ \varepsilon^\xi_{vis}, \varepsilon^\eta_{vis}, \varepsilon^\zeta_{vis} \right]. \quad (2.119) \]

Unfortunately, numerical testing of this viscous preconditioner showed that it was not robust enough to properly handle varying grid aspect ratios. The reasons are not known but the viscous parameter did not equalize the cell crossing times for the pressure perturbations and the pressure waves did not propagate smoothly. Although the same viscous time step is used, the viscous parameter from Buelow [103] is ultimately used in this work. The viscous parameter is given as,

\[ \varepsilon_{vis}^L = \frac{\CFL \nu \frac{M^2}{VNN} \left( \frac{\CFL \nu}{VNN} - \Re \right)}{|\nabla L|^2 \Re^2 \Delta L + M^2 \Re \Delta L \left( \frac{\CFL \nu}{VNN} - \Re \right)} \quad (2.120) \]

where the cell Reynolds number is defined as,

\[ \Re = \frac{\rho U}{\mu} \quad (2.121) \]

A limiter is used for the viscous parameter so that if it is zero it is set to \( \varepsilon_{\text{lim}}^2 \).

CHOPA is written in such a manner that it can automatically select whether the inviscid or viscous preconditioner should be applied at each grid point based on local flow parameters. By using the maximum of the inviscid (Equation (2.112)) or viscous (Equation (2.119)) preconditioning parameter the convergence rate will be optimized,

\[ \varepsilon = \max \left[ \varepsilon_{\text{inv}}, \varepsilon_{\text{vis}} \right]. \quad (2.122) \]

### 2.4.5 Boundary Conditions

Researchers in the field of preconditioners have observed that to keep the accuracy of the original discretization method, characteristic based boundary conditions and artificial dissipation terms must be rederived for the preconditioned equations. This is because the preconditioners alter the eigenvalues, which are the basis for characteristic based boundary conditions [104, 114]. The use of a characteristic boundary condition based on the original system of equations in a preconditioned system can result in spurious reflections of waves from boundaries and is often numerically unstable [184, 185].
2.4.5.1 Method of Characteristics

The Method of Characteristics was initially developed by Thompson [186] with a later modification for viscous flows by Poinsot and Lele [187]. It is a method that can be used for most types of boundaries (e.g. inflow, outflow, walls) by identifying what information is exiting or entering the domain and then either calculating or specifying the information at the boundary based on the direction of propagation. This direction is known by examining the characteristics of the flow. For example, in two-dimensional subsonic flow there are three characteristics that propagate with the flow (two related to the momentum equations and one from the continuity equation for the acoustic speed), \(u\), \(u\), and \(u + c\), and one that propagates against the flow, \(u - c\). The discussion that follows will assume a basic flow that travels in the \(+x\)-direction. So, at a subsonic inflow, there are three waves that have information entering the domain, while one wave is leaving; thus, at the inflow boundary three properties are specified and one is extrapolated from information within the domain. Likewise, at a subsonic outflow there are three characteristics leaving the domain, which are calculated using values from within the domain, and one that is specified (i.e. it enters the domain). More information on the application and derivation of Method of Characteristic boundary conditions can be found in References [184, 188].

The preconditioned Method of Characteristics can be described mathematically as follows. It is the same methodology as the unpreconditioned equations. If the computational domain is properly designed there should not be significant viscous effects at an inflow or an outflow boundary and the Euler equations are suitable to impose the boundary conditions. The three-dimensional preconditioned Euler equations are written as,

\[
\Gamma \frac{\partial Q_v}{\partial t} + \frac{\partial E_1}{\partial \xi} + \frac{\partial F_1}{\partial \eta} + \frac{\partial G_1}{\partial \zeta} = H
\]  

(2.123)

This is the same as Equation (2.22) but \(E_1\), \(F_1\), and \(G_1\) are only composed of the inviscid fluxes. The fluxes are expanded to highlight the coordinate transformation and underscores that this method will work regardless of the solution variable used. Rewriting Equation (2.123) and assuming no sources,
\[ \frac{\partial Q_v}{\partial t} + \Gamma^{-1} \left\{ A_v \frac{\partial Q_v}{\partial \xi} + B_v \frac{\partial Q_v}{\partial \eta} + C_v \frac{\partial Q_v}{\partial \zeta} + M_\xi + M_\eta + M_\zeta \right\} = 0. \] (2.124)

Examination of the \( M \) matrices show they are independent of the solution set used. If the grid is properly constructed then the first derivative will be continuous across the grid and each \( M \) matrix will be zero. These terms are provided here,

\[
M_\xi = E \left( \frac{\xi_x}{J} \right) + F \left( \frac{\xi_y}{J} \right) + G \left( \frac{\xi_z}{J} \right) \xi 
\]

(2.125)

\[
M_\eta = E \left( \frac{\eta_x}{J} \right) + F \left( \frac{\eta_y}{J} \right) + G \left( \frac{\eta_z}{J} \right) \eta 
\]

(2.126)

\[
M_\zeta = E \left( \frac{\zeta_x}{J} \right) + F \left( \frac{\zeta_y}{J} \right) + G \left( \frac{\zeta_z}{J} \right) \zeta 
\]

(2.127)

Since the boundary condition is only used for the derivative normal to the boundary the terms that are not normal are treated as a constant, \( C \), and are calculated from the Navier-Stokes equation during the normal solution procedure. The equations are rewritten for an arbitrary boundary with the use of a generic flux Jacobian,

\[ \frac{\partial Q_v}{\partial t} + \Gamma^{-1} D_v \frac{\partial Q_v}{\partial L} + \Gamma^{-1} C = 0. \] (2.128)

Where the generic flux Jacobian, \( D_v \), depends on the direction of interest and is determined by \( L \). When \( L \) is replaced by the direction normal to the boundary \( D_v \) becomes,

\[
L = \xi \text{ then } D_v = A_v 
\]

(2.129)

\[
L = \eta \text{ then } D_v = B_v 
\]

(2.130)

\[
L = \zeta \text{ then } D_v = C_v. 
\]

(2.131)
The subscript on $L$ is an indication of a derivative with respect to that subscript. When calculating the boundary condition for the other coordinate directions reselect the $L$ term and recalculate the components. The other variables in Equation (2.132) are, total enthalpy, $h_0 = (E_t + \frac{p}{\rho})$; $q^2 = u^2 + v^2 + w^2$; and $U$ is a generic contravariant velocity,

$$U = L_x u + L_y v + L_z w.$$  \hspace{1cm} (2.133)

The matrices $\Gamma^{-1}$ and $D_v$ can be multiplied together to form $D_{\text{prec}}$. This matrix is then expanded into matrices of its left eigenvector, $S^{-1}$, right eigenvector, $S$, and eigenvalues, $\Lambda$.

$$\frac{\partial Q_v}{\partial t} + S\Lambda S^{-1} \frac{\partial Q_v}{\partial L} + \Gamma^{-1}C = 0. \hspace{1cm} (2.134)$$

For the specification of a boundary condition this is rewritten as,

$$\frac{\partial Q_v}{\partial t} + S\mathcal{L} + \Gamma^{-1}C = 0 \hspace{1cm} (2.135)$$

where $\mathcal{L} = \Lambda S^{-1} \frac{\partial Q_v}{\partial L}$. To simplify numerical implementation this is often further reduced to,

$$\frac{\partial Q_v}{\partial t} + d + \Gamma^{-1}C = 0. \hspace{1cm} (2.136)$$

Where $d = S\mathcal{L}$ and $\mathcal{L}$ is specified through the boundary condition. If the charac-
teristic information is entering the domain then that element of $\mathcal{L}$ is specified with a value depending on the boundary condition being simulated. However, if the characteristic information is exiting the computational domain then that element of $\mathcal{L}$ is calculated using one-sided stencils within the domain.

The terms in the two-dimensional preconditioned Method of Characteristics boundary conditions are as follows.

$$\Lambda = \begin{cases} 
\lambda_1 & \frac{1}{2} [(1 + \varepsilon)U - C'] \\
\lambda_2 & U \\
\lambda_3 & U \\
\lambda_4 & \frac{1}{2} [(1 + \varepsilon)U + C'] 
\end{cases} \quad (2.137)$$

$$\mathcal{L} = \begin{cases} 
\frac{\lambda_2}{A_1} \left\{ [\nabla L]^2 (A_2 + C') dp - \rho [A_2 (A_2 + C') + A_3] (L_x du + L_y dv) \right\} \\
\frac{\lambda_2}{\varepsilon L} \left\{ -\frac{1}{\rho} dp + \frac{A_2 L_x}{\nabla L^2} du + \frac{A_2 L_y}{\nabla L^2} dv + \frac{e^{\gamma - 1}}{\varepsilon L} dt \right\} \\
\frac{\lambda_3}{\nabla L^2} \left\{ -L_y du + L_x dv \right\} \\
\frac{\lambda_4}{A_1} \left\{ -[\nabla L]^2 (A_2 - C') dp + \rho [A_2 (A_2 - C') + A_3] (L_x du + L_y dv) \right\} 
\end{cases} \quad (2.138)$$

$$d = \begin{cases} 
\mathcal{L}_1 \rho [A_2 (A_2 - C') + A_3] + \mathcal{L}_4 \rho [A_2 (A_2 + C') + A_3] \\
\mathcal{L}_1 L_x (A_2 - C') - \mathcal{L}_3 L_y + \mathcal{L}_4 L_x (A_2 + C') \\
\mathcal{L}_1 L_y (A_2 - C') + \mathcal{L}_3 L_x + \mathcal{L}_4 L_y (A_2 + C') \\
2(\mathcal{L}_1 + \mathcal{L}_4) (\gamma - 1) [\nabla L]^2 T + \mathcal{L}_2 
\end{cases} \quad (2.139)$$

Where, the subscripts on $\mathcal{L}$ and $\lambda$ denote the elements of the vectors and,
\[
|\nabla L|^2 = L_x^2 + L_y^2 \\
A_1 = 4c^2 \varepsilon \rho C' (|\nabla L|^2)^2 \\
A_2 = (\varepsilon - 1)U \\
A_3 = 2c^2 \varepsilon |\nabla L|^2
\] (2.140)
(2.141)
(2.142)
(2.143)

There are four possible cases for the two-dimensional Method of Characteristics and these are listed below [186,187]. The calculation of \( L \) is such that \( L_1 \) is for the continuity equation, \( L_2 \) and \( L_3 \) are for the momentum equations, and \( L_4 \) is for the energy equation. The CHOPA code is set up to handle flow in arbitrary directions, but the example below assumes that the flow is traveling in the +\( x \)-direction.

**Left Boundary**

\[
\mathcal{L}_{\text{inflow}} = \begin{cases} 
L_1 \\
0 \\
0 \\
0 
\end{cases} \\
\mathcal{L}_{\text{outflow}} = \begin{cases} 
L_1 \\
L_2 \\
L_3 \\
\sigma_{\text{out}} (1 - M^2)(p - p_{\infty}) 
\end{cases}
\] (2.144)

**Right Boundary**

\[
\mathcal{L}_{\text{inflow}} = \begin{cases} 
0 \\
0 \\
0 \\
L_4 
\end{cases} \\
\mathcal{L}_{\text{outflow}} = \begin{cases} 
\sigma_{\text{out}} (1 - M^2)(p - p_{\infty}) \\
L_2 \\
L_3 \\
L_4 
\end{cases}
\] (2.145)
The values for some elements are set to zero to prevent incoming vorticity or entropy. The $\sigma_{\text{out}}$ is defined in Equation (2.146) depending on if it is inviscid or viscous flow and assuming that there are no body forces. Lele and Poinset [187] found it helpful to add a pressure correction for viscous flows to Thompson’s specification of the incoming pressure wave. They found that if the pressure at the outlet did not match the freestream pressure there were often small reflections.

\[
\begin{align*}
\text{inviscid} & : \quad \sigma_{\text{out}} = 0 \\
\text{viscous} & : \quad \sigma_{\text{out}} = \frac{1}{4}
\end{align*}
\]  

(2.146)

Because of the way the Method of Characteristics boundary condition is defined, the primitive variables are not set at the boundary, they are rather based on perturbations in the flow. Because of this, at an inflow boundary, the incoming velocity will tend to fluctuate about the mean flow. This highlights why the Method of Characteristics boundary condition is not recommended for the inflow boundary.

### 2.4.5.2 Extrapolation Boundary Condition

While the Method of Characteristics boundary condition is robust it is also mathematically complex [186, 187]. Since the preconditioner affects the characteristics of the flow a new Method of Characteristics boundary condition would need to be developed for each preconditioner. As an alternative an extrapolation boundary condition is used. In conjunction with a buffer zone the extrapolation boundary condition is an effective non-reflecting boundary condition.

The extrapolation boundary condition follows the same mathematical principles as the Method of Characteristics. Each equation has a wave associated with it and information can be gleaned on the direction of travel: acoustic waves for the continuity equation, entropy waves for the momentum equations, and vorticity waves for the energy equation. The contravariant velocity is used to determine if the information is entering or leaving the computation domain. For waves that are entering the domain the conservative variable corresponding to that wave is set to the freestream value. Waves that are exiting the domain use a three point extrapolation to determine the value of the variable on the boundary. This is shown in Equation (2.147) for a variable on the boundary at $x = i$. 

70
\[ f_i = 3f_{i-1} - 3f_{i-2} + f_{i-3} \]  

Thus for the subsonic flow that is the focus of this work at an inflow boundary condition the density (e.g. continuity equation) is calculated and the momentum and total energy is prescribed from freestream values. As an outflow boundary the incoming density is set to the freestream value while the momentum and total energy on the boundary are calculated from the interior information. The buffer zone helps to damp the flow field to prescribed values, simplifying the application of the extrapolation boundary condition.

### 2.4.5.3 Buffer Zone

A buffer zone is used in conjunction with grid stretching and an extrapolation boundary condition to provide an accurate non-reflecting boundary condition. The buffer zone gradually damps the solution to a prescribed value (e.g. the freestream value) as a boundary is approached. By forcing the solution to approach a uniform value at the boundaries this boundary condition is akin to a nonreflecting boundary condition. In Figure 2.2 the buffer zone is shown in smooth contours from 0 to 0.1 (blue to red, respectively). A slice through this plot at \( y = 0 \) is shown in Figure 2.3. These plots show how the buffer zone is exponentially increased as the boundary is approached, increasing the damping as the disturbance approaches the boundary.

This method is much less numerically intensive than a Method of Characteristics boundary condition. The buffer zone can be applied in such a manner as to be applicable for both the unpreconditioned and preconditioned versions of the code. Two buffer zone methods are examined in this work and the merits of each are discussed below.

#### 2.4.5.3.1 WGK Buffer Zone

The first buffer zone examined was developed by Wasistho, Geurts, and Kuerten (WGK) [189] for enhancing the stability of other boundary conditions used in their computational code. The method was applied to flow instability problems that used a Riemann boundary condition for the inflow and a Method of Characteristic boundary condition for the outflow. In this manner, the numerical stability of their analyses was increased to allow large flow instabilities to exit the domain smoothly.
Figure 2.2. Buffer zone applied to all boundaries.

Figure 2.3. Plot of JBF buffer zone along $y = 0$. $F_1$ is the magnitude of the damping function.

The WGK method works by damping the solution variables towards reference values, which are typically the freestream variables. The solution vector, $Q$, is adjusted within the buffer zone as,

$$Q = Q_\infty + \varsigma (Q - Q_\infty).$$  \hspace{1cm} (2.148)

Here $\varsigma$ is the magnitude of damping in the buffer zone and is given by Equation (2.149).

$$\varsigma = (1 - C_1 x_b^2) \left(1 - \frac{1 - \exp(C_2 x_b^2)}{1 - \exp(C_2)}\right)$$  \hspace{1cm} (2.149)

Where $C_1$ and $C_2$ are coefficients that control the shape of the damping function. Wasistho, Geurts, and Kuerten suggest values of $C_1 = 0.0$ and $C_2 = 7.0$, which are used in this analysis. The final variable, $x_b$, is defined as,

$$x_b = \frac{x_i - x_{\text{start}}}{x_{\text{end}} - x_{\text{start}}}.\hspace{1cm} (2.150)$$

Where the current location within the buffer zone is $x_i$ and the beginning and end locations of the buffer zone are given by $x_{\text{start}}$ and $x_{\text{end}}$, respectively.

The WGK buffer zone method is not suggested for implementation. From numerical testing the method works well for steady state problems but not for
time accurate problems. As the solution variables are damped directly with this method, the mathematical validity of the governing equations in the buffer region is removed. This results in a nonphysical residual for the buffer zone, which must be excluded when calculating the residual for convergence criterion. However, when a time accurate solution is calculated, by not including the buffer zone in the convergence criterion the solution can be converged in the interior of the domain, but not in the buffer zone (e.g. disturbances have not propagated completely out of the domain). Exclusion of these terms when checking for convergence results in premature convergence, which eventually leads to an unstable solution. The second buffer zone method addresses this issue by damping the residual and maintaining mathematical integrity across the domain.

2.4.5.3.2 JBF Buffer Zone The second buffer zone method was formulated by Freund and is referred to as JBF [190]. The premise behind this buffer zone is very similar to the WGK condition except that the damping term acts upon the residual instead of the solution. The buffer zone is defined in the same manner as before, with the damping increasing towards the boundary based on a user defined exponent. Although the interior of the domain for the WGK had a damping value of one the JBF method uses a damping value of zero to indicate the non-buffer zone region.

The JBF method works by adding an artificial convective velocity and damping term to each of the governing equations (e.g. continuity, momentum, and energy). For simplicity this will be presented just for the continuity equation in the Euler equations, but the other equations are similar. This is also effective for the Navier-Stokes equations and the extension is straightforward. The Euler equations with a buffer zone treatment can be written in vector form as,

$$\frac{\partial \mathbf{Q}}{\partial t} = - \frac{\partial \mathbf{E}_i}{\partial x_i} - \mathbf{U}_\infty \frac{\partial \mathbf{E}_i}{\partial x} + \sigma(x)(\mathbf{Q} - \mathbf{Q}_\infty).$$  (2.151)

Where $\mathbf{U}_\infty$ is the mean velocity in the $x$-direction and is an artificial convective term used to accelerate the flow in the buffer zone to supersonic speed, where it is trivial to implement characteristic type boundary conditions since all information is either entering or leaving the domain. However, since all work with this method will be for very low Mach number flows, the direction of the characteristics is the
same for all problems and this term can be omitted. Additionally, since the preconditioner changes the speed of sound an additional change within the buffer zone may introduce numerical instabilities. The $\sigma$ term is the amplitude of damping in the buffer zone, which changes depending on the location in the buffer zone and is given by,

$$\sigma(x) = \overline{\sigma} \left( \frac{x_i - x_{\text{end}}}{x_{\text{start}} - x_{\text{end}}} \right)^{\beta}. \quad (2.152)$$

The $\overline{\sigma}$ in Equation (2.152) is the maximum amplitude of the damping in the buffer zone and is usually set to a value corresponding to the freestream speed of sound. This value is sensitive to whether the code is run in preconditioned or unpreconditioned form. Typically, the preconditioned value is one to two orders of magnitude smaller than the freestream speed of sound. The spatial variation of the damping is determined by $\beta$ and will be discussed further in Section 3.1.2.

A difficulty with this method is that it requires a reference value, which is typically the freestream value. For the outflow boundary where the flow is over a viscous wall there is some difficulty in successfully applying the JBF method. Since the boundary layer will grow in thickness as the flow evolves downstream a nonconstant buffer zone would be required. However, as the solution in the buffer zone is nonphysical the boundary layer profile is frozen in the buffer zone and corresponds to the thickness at the point where the buffer zone starts. Also, grid stretching is applied to the domain near the boundaries, further rendering the solution non-physical in the buffer zone region. This thickness can be determined in one of three ways. The first involves solving a separate problem with a larger domain to determine the exact boundary layer thickness at that point, which is impractical. The second method is to approximate the boundary layer thickness at that point using relationships given by White [191]. There could be some problems with this method as the interface plane between the buffer zone and physical domain has a specified solution but this is only an approximation for the flow. This could lead to a discontinuity in the flow and possibly numerical instability.

The third method is a transitional buffer zone that is used perpendicular to viscous walls, with the reference value used in the solution dynamically mimicking the solution at the last interior point. Grid stretching is used in conjuction with
the buffer zone. This has been used with success in jet noise simulations [192] and is selected for use.

An additional advantage with JBF is that the buffer zone can be coupled with a characteristic type boundary condition to enforce a mean flow parameter at the boundary. When using a Method of Characteristics boundary condition it is not possible to exactly specify the mean flow value of pressure (or density) at the boundary, so it typically wanders around the freestream value. Specifying the outflow pressure is very important in pipe flow simulations and the buffer zone could be used to overcome this problem. This advantage has been recently addressed by Hixon with success [193] and should be reexamined for future application.

2.4.6 Artificial Dissipation

Artificial dissipation must be added to spatial discretizations that are calculated with central difference schemes. It is required to damp spurious numerical oscillations. The dissipation is usually a second or fourth order difference term, with second order used near shocks and fourth order for high frequency errors. For the preconditioned equations the preconditioner should not be applied to the artificial dissipation terms. However, if a matrix based dissipation is used it should be based on the preconditioned equations [107, 114, 194].

The low speed flows encountered with underwater vehicles and automobiles has special problems associated with the artificial dissipation. Darmofal and van Leer [155] and Venkateswaran and Merkle [195] have shown that poorly balanced artificial dissipation matrices exist as the Mach number approaches zero. In numerical simulations that use the much more common scalar artificial dissipation the dissipation is usually scaled by the spectral radius of the inviscid flux Jacobian. So, at very low Mach numbers, this results in too much dissipation for the continuity equation with too little dissipation for the momentum and energy equations [108, 196]. This is because the artificial dissipation coefficient for the continuity equation scales with $M$ while, in the momentum and energy equations it scales with $M^{-1}$. An additional concern that is related to round-off errors is that in very low Mach number flows the density is almost constant, hence the artificial dissipation should be based on the pressure and not the density [107]. The over-damping with scalar artificial dissipation is a known problem for low Mach number
flows and was first documented by Volpe [181]. In his work he compared the accuracy of numerical simulations to experimental data as the Mach number was reduced. The reduction in accuracy was notable once the Mach number dropped below $M = 0.01$ for the pressure distribution across an inviscid cylinder. The derivation of a matrix-based artificial dissipation will now be discussed.

This is similar to an upwind differencing method and provides good results at very low Mach numbers. The difference between matrix artificial dissipation and scalar artificial dissipation is the scaling coefficient for the dissipation. With a matrix-based artificial dissipation, the eigenvalue of the inviscid flux Jacobian that goes with each equation is used as the scaling coefficient as opposed to the spectral radius. This results in the correct amount of dissipation and accurate solutions at low Mach numbers. This technique can be applied to the preconditioned or unpreconditioned version of the code, but since each version has different eigenvalues and eigenvectors (due to the preconditioning) it is only developed for the preconditioned equations. The basis of the matrix artificial dissipation can be found in Swanson and Turkel [197] and Turkel and Vatsa [198].

The preconditioned Navier-Stokes equations, in nondimensional, vector form for a generalized coordinate system can be written as,

$$\Gamma \frac{\partial Q_v}{\partial t} + \frac{\partial E_1}{\partial \xi} + \frac{\partial F_1}{\partial \eta} + \frac{\partial G_1}{\partial \zeta} = H \tag{2.153}$$

where $\Gamma$ is the CMv-B preconditioner. Including the artificial dissipation, $AD$, and solving for the time derivative results in,

$$\frac{\partial Q_v}{\partial t} = -\Gamma^{-1} \left[ \frac{\partial E_1}{\partial \xi} + \frac{\partial F_1}{\partial \eta} + \frac{\partial G_1}{\partial \zeta} + H \right] + AD. \tag{2.154}$$

The preconditioned convective fluxes should be used to determine the eigenvalues. Only the convective terms are included when calculating the eigenvalues as the matrix dissipation tries to mimic an upwind discretization scheme, which only upwinds the convective terms. The viscous terms are typically not upwinded as they inherently possess dissipation.

The equations for the artificial dissipation can be rewritten with flux Jacobians by dropping the viscous terms,
\[
\frac{\partial Q_v}{\partial t} = -\Gamma^{-1} \left[ A_v \frac{\partial Q_v}{\partial \xi} + B_v \frac{\partial Q_v}{\partial \eta} + C_v \frac{\partial Q_v}{\partial \zeta} + H \right] + AD. \tag{2.155}
\]

The flux Jacobians, \(A_v, B_v, C_v\), can be represented with a generic matrix as in Equation (2.132). For matrix artificial dissipation the artificial dissipation term is divided into different derivatives and coordinate directions is written as,

\[
AD = \left( D^2_\xi + D^2_\eta - D^4_\xi - D^4_\eta - D^4_\zeta \right) Q_v. \tag{2.156}
\]

The superscript refers to the order of the derivative and the subscript refers to the coordinate direction. Second order dissipation is only required for the capturing of shock waves (due to the sharp discontinuity in field variables on opposite sides of the shock) and will be omitted. The fourth derivative dissipation is written for point \((i, j)\) as,

\[
D^4_\xi Q_v = \nabla_\xi \left[ \left( |\Gamma^{-1} A_v|_{i+1/2,j} \right) \Delta_\xi \nabla_\xi \Delta_\zeta \right] Q_v \tag{2.157}
\]

\[
D^4_\eta Q_v = \nabla_\eta \left[ \left( |\Gamma^{-1} B_v|_{i+1/2,j} \right) \Delta_\eta \nabla_\eta \Delta_\eta \right] Q_v \tag{2.158}
\]

\[
D^4_\zeta Q_v = \nabla_\zeta \left[ \left( |\Gamma^{-1} C_v|_{i+1/2,j} \right) \Delta_\zeta \nabla_\zeta \Delta_\zeta \right] Q_v \tag{2.159}
\]

The half-point \((i + 1/2)\) values are calculated using a simple arithmetic average over the \(i\) and \(i+1\) points. \(\nabla\) and \(\Delta\) are the standard, first order accurate, forward and backward difference operators for each direction,

\[
\nabla Q_{i,j} = Q_{i,j} - Q_{i-1,j} \tag{2.160}
\]

\[
\Delta Q_{i,j} = Q_{i+1,j} - Q_{i,j}. \tag{2.161}
\]

As it is derived here, the matrix dissipation is formally second order accurate. This is because the original derivation of the matrix terms was based on a second order accurate upwind difference approximation for the spatial derivations [199]. Since an upwind scheme has dissipation inherent in its formulation the upwind scheme is reconstituted into a second order central difference and a fourth derivative dissipation.
The coefficient $\epsilon^4$ scales on flow parameters and is based on Lockard and Morris’s paper on scalar artificial dissipation [98],

$$\epsilon^4 = \max [d_6, (d_6 + \phi)]. \tag{2.162}$$

The magnitude of the background dissipation is given in $d_6$ and is the minimum level that is continuously added to the governing equations. The other term is defined for the $\xi$ direction (other directions are similar) as,

$$\phi = \chi \beta_1 k_{\text{med}} \tag{2.163}$$

$$k_{\text{med}} = \max (0, 0.01 - d_6) \tag{2.164}$$

$$\chi = \min [\beta_0, \max (0.0, \nu_{\text{max}} - d_2)] \tag{2.165}$$

$$\nu_{\text{max}} = \max [\nu_{i-1}, \nu_i, \nu_{i+1}] \tag{2.166}$$

$$\nu_i = \min \left( k_2, \frac{\kappa \partial^4 p_i}{\partial \xi^4} \right) \tag{2.167}$$

$$k_2 = 0.25 k_{\text{max}} + \nu_2 \tag{2.168}$$

$$\beta_0 = 0.1 \nu_2 - d_2 \tag{2.169}$$

$$\beta_1 = \max [1 \times 10^{-6}, 1/\beta_0] \tag{2.170}$$

$$d_2 = 0. \tag{2.171}$$

where $p_i$ is pressure. It should be noted that $\beta_0$ and $\beta_1$ are reversed in the original definitions of $\phi$ and $\chi$ as numerical experiments have found it to be more stable. $\kappa$ is a scaling coefficient that is set to 0.35 and $k_{\text{max}} = 0.15$. The definition of $k_2$ has been modified; if the second derivative dissipation were being used, $\nu_2$ defines a threshold value ($\sim 0.02$) below which no second-order dissipation is used.

To calculate the eigenvalue scaling factors first expand the preconditioned flux Jacobians (dropping the $i + 1/2, j$ subscripts and using the general formulation of $D_\nu$ for clarity),

$$\left| \Gamma^{-1} D_\nu \right| = S_R |\Lambda_V| S_L^{-1}. \tag{2.172}$$

Where, $S_R$ and $S_L^{-1}$ are the right and left eigenvectors, respectively, and $\Lambda_V$ is the vector of eigenvalues. These have been derived for the second CMv-B preconditioner, $\Gamma_2$, using Mathematica. This matrix is given in two dimensions (with the use
of column vectors, $S_i$ for readability) as,

$$S_R|\Lambda|S_L^{-1} = \frac{1}{4G_2^2\rho L^2 C'} \left[ \begin{array}{c} \{S_1\} \{S_2\} \{S_3\} \{S_4\} \end{array} \right]. \quad (2.173)$$

The following substitutions are used:

$$S_1 = \begin{bmatrix} \rho L^2 (\lambda_1 + H_3) + H_4 \lambda_3 \\ -L_x \overline{L}^2(\lambda_3 - \lambda_4) [(G_1^2 + 2G_2)^2 - (G_1 C')^2] \\ -L_y \overline{L}^2(\lambda_3 - \lambda_4) [(G_1^2 + 2G_2)^2 - (G_1 C')^2] \\ -(\gamma - 1)L \overline{L}^2 [H_1 \lambda_3 (G_1 - C') + 4G_2 \lambda_1 C' - H_5 \lambda_4 (G_1 + C')] \end{bmatrix},$$

$$S_2 = \begin{bmatrix} -H_3 H_4 L_x \rho^2 (\lambda_3 - \lambda_4) \\ \rho \left( H_3 H_5 \lambda_3 + H_1 H_4 \lambda_4 + 4G_2^2 L_2 \lambda_2 C' \right) \\ L_x \lambda_4 \rho \left( H_3 H_5 \lambda_3 + H_1 H_4 \lambda_4 - 4G_2^2 \lambda_2 C' \right) \\ (\gamma - 1)L_x \rho \overline{L}^2 [4G_1 G_2 \lambda_1 C' - H_3 \lambda_3 (C' - G_1) + H_4 \lambda_4 (G_1 + C')] \end{bmatrix},$$

$$S_3 = \begin{bmatrix} -H_3 H_4 L_y \rho^2 (\lambda_3 - \lambda_4) \\ L_x \lambda_4 \rho \left( H_3 H_5 \lambda_3 + H_1 H_4 \lambda_4 - 4G_2^2 \lambda_2 C' \right) \\ \rho \left( H_3 H_5 \lambda_3 + H_1 H_4 \lambda_4 + 4G_2^2 L_2 \lambda_2 C' \right) \\ (\gamma - 1)L_y \rho \overline{L}^2 [4G_1 G_2 \lambda_1 C' - H_3 \lambda_3 (C' - G_1) + H_4 \lambda_4 (G_1 + C')] \end{bmatrix},$$

$$S_4 = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 4G_2^2 \rho \overline{L}^2 \lambda_1 C' \end{bmatrix}.$$

Additional variable definitions are,

$$C' = \sqrt{4c^2 \varepsilon L^2 + (\varepsilon - 1)^2 U^2} \quad (2.174)$$

$$H_1 = 2G^2 + G_1 (G_1 + C') \quad (2.175)$$

79
\[ H_2 = 2G^2 - G_1(G_1 + C') \]  
\[ H_3 = G_1^2(G_1 + C') + G_2(3G_1 + C') \]  
\[ H_4 = G_1^2(-G_1 + C') + G_2(-3G_1 + C') \]  
\[ G_1 = (\varepsilon - 1)U \]  
\[ G_2 = c^2\varepsilon L^2 \]  
\[ L^2 = L_x^2 + L_y^2 \]

Again, \( L \) is used to select between coordinate directions (e.g. \( L = \eta \), etc.). The preconditioned eigenvalues given by

\[ \Lambda_v = \begin{pmatrix} \lambda_1 \\ \lambda_2 \\ \lambda_3 \\ \lambda_4 \end{pmatrix} = \begin{pmatrix} |U| \\ |U| \\ \frac{1}{2} |U(1 + \varepsilon)| - C' \\ \frac{1}{2} |U(1 + \varepsilon)| + C' \end{pmatrix} \]

However, these eigenvalues can introduce numerical instability near stagnation points since they will approach zero. Thus they are modified as,

\[ \Lambda_v = \begin{pmatrix} \tilde{\lambda}_1 \\ \tilde{\lambda}_2 \\ \tilde{\lambda}_3 \\ \tilde{\lambda}_4 \end{pmatrix} = \begin{pmatrix} \max(\lambda_1, \lambda_2, V_l\lambda_4) \\ \max(\lambda_1, \lambda_2, V_l\lambda_4) \\ \max(\lambda_3, V_n\lambda_4) \\ \lambda_4 \end{pmatrix} \]

Typically, the entropy/vorticity limiter, \( V_l \), and the acoustic limiter, \( V_n \), are both set to zero for subsonic flow and \( V_l = 0.025 \), \( V_n = 0.25 \) for transonic flow [197, 200]. The limiters are designed to impose a maximum of one quarter of the acoustic eigenvalue for the convective eigenvalues, but for preconditioning the acoustic and convective eigenvalues are approximately equal. Thus, the limiters designed for transonic flow are used.

The final modification to the eigenvalues are for viscous computations, which require very fine grid spacings normal to the wall. These simulations result in
high aspect ratio grids, which can result in too much artificial dissipation being produced [197, 198]. As a result, the eigenvalue for one coordinate direction is scaled based on the eigenvalues in all directions. The scaled eigenvalues replace those in Equation (2.183) and are given for any eigenvalue \((\lambda_i, i = 1 \text{ to } 4)\) in generalized coordinates by

\[
\lambda_\xi = \tilde{\lambda}_\xi \left[ 1 + \left( \frac{\tilde{\lambda}_\eta}{\lambda_\xi} \right)^{1/2} + \left( \frac{\tilde{\lambda}_\zeta}{\lambda_\xi} \right)^{1/2} \right] \tag{2.184}
\]

\[
\lambda_\eta = \tilde{\lambda}_\eta \left[ 1 + \left( \frac{\tilde{\lambda}_\xi}{\lambda_\eta} \right)^{1/2} + \left( \frac{\tilde{\lambda}_\zeta}{\lambda_\eta} \right)^{1/2} \right] \tag{2.185}
\]

\[
\lambda_\zeta = \tilde{\lambda}_\zeta \left[ 1 + \left( \frac{\tilde{\lambda}_\xi}{\lambda_\zeta} \right)^{1/2} + \left( \frac{\tilde{\lambda}_\eta}{\lambda_\zeta} \right)^{1/2} \right] \tag{2.186}
\]

Finally, Equation (2.157) will be expanded to show the numerical implementation. This is for \(\xi\) only but other directions are similar. By following the rules for forward and backward differencing the equation results in (dropping the \(v\) subscript and \(j\) position subscript for clarity),

\[
D_\xi^4 Q_i = (\psi_{i-1/2} - 3\psi_{i-1/2} + \psi_{i+1/2}) Q_{i-1} + 3 (\psi_{i-1/2} + \psi_{i+1/2}) Q_i - (\psi_{i-1/2} + 3\psi_{i+1/2}) Q_{i+1} + (\psi_{i+1/2}) Q_{i+2} \tag{2.187}
\]

where \(\psi_i = |\Gamma^{-1} A_v|, \varepsilon_i^4\).

Because the matrix artificial dissipation uses a five point stencil, a biased stencil is used at the first interior point from a boundary. Remember that no dissipation is applied on the boundary as it is handled by the boundary condition).

\(imin+1:\)

\[
D_\xi^4 Q_i = 2 (\psi_{i-1/2}) Q_{i-1} - (3\psi_{i-1/2} + 2\psi_{i+1/2}) Q_i + (\psi_{i-1/2} + 3\psi_{i+1/2}) Q_{i+1} - (\psi_{i+1/2}) Q_{i+2} \tag{2.188}
\]

\(imax-1:\)
\[ \mathcal{P}_\xi^i Q_i = - \left( \psi_{i-1/2} \right) Q_{i-2} + \left( 3 \psi_{i-1/2} + \psi_{i+1/2} \right) Q_{i-1} - \left( 2 \psi_{i-1/2} + 3 \psi_{i+1/2} \right) Q_i + 2 \left( \psi_{i+1/2} \right) Q_{i+1} \quad (2.189) \]

The final piece for the matrix artificial dissipation is that CHOPA is written using the conservative variable set, \( Q = [\rho, \rho u, \rho v, \rho w, \rho E_t]^T \) but the artificial dissipation is derived for the \( Q_v \) variable set. Turkel and Vatsa [201] show that the residual will often plateau before convergence when the conservative variable set is used for the artificial dissipation. This phenomenon is explained by realizing that the density (i.e. conservative variables) is approximately constant for a low Mach number flow, while the pressure is not. Turkel and Vatsa have also shown that the convergence history is independent of the variables used to integrate in time. On a side note, the derivation was performed in this manner as it was too difficult to symbolically calculate the eigenvectors and eigenvalues for any other variable set. Thus, to keep the equation set mathematically consistent, Equation (2.154) must be multiplied by the Jacobian \( \partial Q/\partial Q_v \), giving,

\[ \frac{\partial Q}{\partial t} = - \frac{\partial Q}{\partial Q_v} \left\{ \Gamma^{-1} \left[ \frac{\partial E_1}{\partial \xi} + \frac{\partial F_1}{\partial \eta} + \frac{\partial G_1}{\partial \zeta} + H \right] + AD \right\}. \quad (2.190) \]

### 2.5 Turbulence Modeling

Since the simulation of most flows will be turbulent, a turbulence model must be included in the system of equations. When a turbulence model is used with the Navier-Stokes equations the preconditioner must now balance the turbulent sources in addition to convection and diffusion of the flow. Algebraic and one-equation turbulence models can be accounted for by including them with the differential Navier-Stokes equations and then applying the preconditioner to the entire system of equations but this is not necessary [147]. A viscous Jacobian preconditioner can also be adapted that has a source term for the turbulence model. Lee [147] has incorporated the Spalart-Allmaras one-equation turbulence model into a Navier-Stokes preconditioner through the source term in the V-VJ97 preconditioner. However, in this work the preconditioner is not applied to the turbulence model.
2.5.1 Spalart-Allmaras One-Equation Model

A turbulence model is required to provide closure to the governing differential equations when solving turbulent flows. This closure is required due to the unknown variables introduced by the Favre averaging process. Because of its applicability with a hybrid RANS/LES turbulence model the Spalart-Allmaras turbulence model is selected [85]. This turbulence model is also good because it is relatively insensitive to freestream values as it is an extension of Baldwin and Barth’s one-equation turbulence model [202]. In the Spalart-Allmaras turbulence model they solve for the turbulent eddy viscosity, $\mu_t$, by solving a transport equation for a working variable, $\hat{\nu}$,

$$
\nu_T = \hat{\nu} f_{v1} \quad (2.191)
$$

or,

$$
\mu_T = \bar{\rho} \hat{\nu} f_{v1} \quad (2.192)
$$

This working variable is used because $\hat{\nu}$ behaves linearly near the wall and does not require the use of a damping function [203]. The nondimensional, Favre-averaged transport equation is then given as,

$$
\frac{\partial \hat{\nu}}{\partial t} + \bar{u}_i \frac{\partial \hat{\nu}}{\partial x_j} = c_{b1} \left[ 1 - f_{t2} \right] \hat{S} \hat{\nu} - \frac{M}{Re} \left[ \frac{c_{w1} f_w}{\kappa^2 f_{t2}} - c_{b1} \right] \left[ \hat{\nu} \right]^2 + \frac{M 1}{Re \sigma} \left[ \frac{\partial}{\partial x_k} \left( \nu + \hat{\nu} \right) \frac{\partial \hat{\nu}}{\partial x_k} \right] + c_{b2} \left[ \frac{\partial \hat{\nu}}{\partial x_k} \right]^2 \quad (2.193)
$$

It should be noted that this is slightly different from other formulations in the literature. First, since we are not concerned with predicting laminar to turbulent transition, the trip term, $f_{t2}$ can be set to zero. The functions in Equation (2.193) are given as,
\[ \chi = \frac{\hat{\nu}}{\nu} \]

\[ f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}} \]

\[ f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}} \]

\[ \hat{S} = f_{v3}S + \frac{M}{Re \kappa^2 d^2} f_{v2} \]

\[ f_w = g \left[ \frac{1 + c_{w3}^6}{g^6 + c_{w3}^6} \right]^{1/6} \]

\[ g = r + c_{w2} (r^6 - r) \]

\[ r = \frac{M}{Re \hat{\nu} \hat{S} \kappa^2 d^2} \]

\[ \kappa = 0.41 \]

Where \( \kappa \) is the von Karman constant and \( d \) is the distance from the grid point of interest to the nearest wall. There are multiple definitions of \( S \) with one recommended by Edwards and Chandra [204] that corrects the singular behavior of \( \hat{S} \) in near wall regions. This alternative is given as,

\[ \hat{S} = \sqrt{S} \left( \frac{1}{\chi} + f_{v1} \right) \]  \hspace{1cm} (2.194)

\[ r = \frac{\tanh \left( \frac{\hat{\nu}}{S \kappa^2 d^2} \right)}{\tanh(1.0)} \] \hspace{1cm} (2.195)

The \( S \) term is the magnitude of the vorticity,

\[ S = |\omega_{ij}| = \sqrt{(2\Omega_{ij})(2\Omega_{ij})} \] \hspace{1cm} (2.196)

\[ \Omega_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{u}_i}{\partial x_j} - \frac{\partial \tilde{u}_j}{\partial x_i} \right) \] \hspace{1cm} (2.197)

There are also limiters used in the calculation of the turbulent eddy viscosity. For more information on these please see Shieh [12]. At a solid wall boundary the working variable \( \hat{\nu} \) is set to zero.

Lastly, to further reduce the number of grid points required to simulate the turbulent flow a wall function can be used. This wall function is used to bridge the gap between the laminar sublayer and the fully turbulent layer via the “Law of the
If a wall function is not used then a low Reynolds number turbulence model is required, where the term low Reynolds number refers to the integration of the turbulence model to the wall. For a wall function the first grid point normal to the wall is typically located at $y^+$ between 30 and 100. Where,

$$y^+ = \frac{yu^*}{\nu}$$

(2.198)

with $y$ the distance normal to the wall, $u^*$ is the wall-friction velocity. The wall-friction velocity is related to the density and wall shear stress as,

$$u^* = \sqrt{\frac{\tau_w}{\rho}}$$

(2.199)

A formal derivation of the wall function used in CHOPA is found in Shieh [12].

### 2.5.2 Detached-Eddy Simulation

A well validated method of modeling turbulence in a CFD simulation is with a hybrid RANS/LES approach. Ideally, cavity simulations would benefit from the use of a Large Eddy Simulation (LES) that could simulate all large scales of turbulence. It would then provide approximations for the small scales of turbulence, which are usually independent of the problem and are easy to simulate. However, at high Reynolds number the LES simulations require an inordinate number of points. Thus, a trade-off is found between the time of simulation and accuracy through the use of both RANS and LES in the same simulation. There are different versions of hybrid RANS/LES models but the most prominent one is DES (Detached Eddy Simulation) [206–208]. There are other types of hybrid RANS/LES turbulence models that are similar to DES: Flow Simulation Methodology [209], Limited Numerical Scheme [210], Very Large Eddy Simulation [211], hybrid RANS/LES [212].

DES works well because it models the small scales of the flow, while directly calculating the large scales. Since the large scales (i.e. a vortex) are responsible for most of the energy in the flow, they are the most important and have a great influence on the solution. The small scales are responsible for most of the energy dissipation in the flow field and are similar for most types of flows. With the DES model a RANS type turbulence model is employed in the near-wall bound-
ary layer and LES is in the detached shear layer. By doing this dynamically the more important parts of the flow field are captured by LES in a somewhat automatic fashion. DES has been very effectively for simulating cavity flows by other researchers [1, 213, 214].

With DES if the distance to the wall is below a specified value then the code operates in RANS mode, otherwise it is in LES mode. The LES mode uses a Smagorinksy-like sub-grid scale model and is primarily in the separated region of the shear layer. It is a Smagorinsky-like LES since the mixing length in the turbulence model is proportional to \( \Delta \) (grid spacing). This happens when \( d \gg \Delta \), enacting the LES portion of the model. Thus, the length scale of the model becomes grid-dependent. Since the turbulence model is highly grid dependent care must be taken to have good grid clustering where it is needed (i.e. shear layer). Forethought must be used so that grid-induced separation does not occur, which is when a highly clustered grid near a solid wall causes the RANS model to switch to LES. Lastly, the DES method is not a zonal method, where sections of the flow field are explicitly tagged as being RANS or LES. However, since the grid spacing does determine the switch between RANS and LES can be considered a quasi-zonal method.

To alter the Spalart-Allmaras model to a DES model all that is required is to replace the length scale, \( d \), in Equation (2.193) with,

\[
\tilde{d} \equiv \min (d, C_{DES} \Delta)
\]

(2.200)

where,

\[
\Delta = \max (\Delta x, \Delta y, \Delta z)
\]

(2.201)

\( C_{DES} \) is set to 0.65 based on calibration with isotropic turbulence [203]. Using a larger value of \( C_{DES} \) can suppress the instability waves in shear layers by increasing the subgrid eddy viscosity in the LES regions [215]. If this value is too high the cavity feedback loop could be suppressed. Also, the higher the numerical accuracy a scheme has, the lower the \( C_{DES} \) value can be, as that scheme can more accurately resolve the smaller eddies [216].

Based on previous work [217] a more stable solution can sometimes be achieved if the DES model is turned off entirely in the LES region. The DES model can be
too dissipative in that region. Vortices within a cavity shear layer were damped in an unphysical manner, eliminating the possibility of a cavity tones. For the cavity simulations in this work the DES model was turned off and a laminar calculation was performed in those regions. This significantly increased the stability of the preconditioned solutions.
Chapter 3  
Code Validation

The previous version of CHOP A has been extensively tested for moderate subsonic cavity and jet flows. This chapter covers the validation of low Mach number flows using preconditioning and matrix dissipation.

3.1 Boundary Conditions

The boundary conditions in CHOP A were modified to properly account for the preconditioned system of equations. A brief description of the existing inflow/outflow boundary conditions, as well as the validation of the new boundary conditions are given below. While the buffer zone used in this work is not a boundary condition in the strict sense it does facilitate the proper implementation of the extrapolation boundary condition and is discussed as well.

3.1.1 Existing Inflow/Outflow Boundary Conditions

3.1.1.1 Riemann Invariant Boundary Condition

The unpreconditioned version of CHOP A uses Riemann invariants as a robust inflow boundary condition. While the Method of Characteristics is used for outflow boundaries (see below for a more in depth discussion) it does not allow an exact specification of primitive variables. For this reason the use of the Method of Characteristics for an inflow boundary condition tends to result in primitive variables that waiver about the freestream value. Thus, the Riemann boundary condition was previously used since it permitted an exact specification of the freestream
conditions at the boundary, allowing for a robust nonreflecting boundary. Unfortunately, a stable Riemann boundary condition for the preconditioned equations does not exist. For the preconditioner developed by Buelow a Riemann boundary condition was derived but through numerical testing it was not stable. A Riemann boundary condition that specifies entropy and stagnation enthalpy at an inflow and pressure at an outflow boundary is predicted to be mathematically stable, but the preconditioned Riemann boundary condition was mathematically intractable for that variable set. Other Riemann boundary conditions based on different variables were examined and shown to be unstable. The reader is referred to Reference [218] for details on the mathematical instability of a preconditioned Riemann boundary condition. Based on the cited reference and the author’s experience via numerical testing the Riemann boundary condition is not suitable for a preconditioned solution. As such, an extrapolation boundary condition is used in conjunction with a buffer zone and grid stretching.

3.1.1.2 Method of Characteristics

The mathematical details of a preconditioned Method of Characteristics boundary condition was previously derived in Section 2.4.5.1. The validity of a preconditioned Method of Characteristics boundary condition has been proven by previous researchers [103, 137] yet the boundary conditioned used here was not numerically stable. The variable set used in this boundary condition is different than other researchers. However, it is not expected that this would be the source of instability though. Due to the difficulties with the boundary condition it will not be used in this research. Instead an extrapolation boundary condition with a buffer zone is recommended for inflow and outflow boundaries.

3.1.2 Extrapolation Boundary Condition with Buffer Zone

The extrapolation boundary condition is teamed with the JBF buffer zone to allow disturbances to smoothly exit the domain. Numerical testing is completed to select the coefficients that will most efficiently permit disturbances to exit the domain. To determine the validity of this boundary condition the propagation of an acoustic pulse out of a domain is simulated. This problem is selected as an example of how the buffer zone smoothly damps outgoing disturbances without introducing
reflections at the boundary or the interior/buffer zone interface. Grid stretching is used at the boundaries to assist the boundary condition.

A square domain, comprised of 251 by 251 grid points, is used for the simulation. A square portion of the domain, centered about the origin, with a nondimensional size of four units by four units, is used for the physical domain. This center domain has 151 evenly spaced grid points in each direction. A buffer zone is created on all sides by adding 50 points in each direction with a grid stretching ratio of 1.05. This results in an overall square domain with a length of 15.51, see Figure 3.1.

![Grid used to validate buffer zone and extrapolation boundary condition. Every other grid point has been omitted for clarity.](image)

**Figure 3.1.** Grid used to validate buffer zone and extrapolation boundary condition. Every other grid point has been omitted for clarity.

The initial conditions for the simulation are zero velocity and a Gaussian pressure distribution of,

\[ p = p_\infty + A \exp \left[ -\frac{\log(2) ((x - x_0)^2 + (y - y_0)^2)}{\alpha^2} \right] \]  

(3.1)

with \( x_0 = y_0 = 0 \), \( \alpha = 0.1 \), and \( A = 0.1 \). Density is calculated from the perfect gas law based on the initial condition for pressure. This analysis is conducted using a dual time integration approach, without preconditioning. The results are compared to an exact solution that has been derived for the linearized Euler equations [86]. A detailed derivation of the exact solution can be found in Reference [219]. In this manner variables in the buffer zone of the validation problem could be compared.
to the exact solution, which simulates propagation in free space.

There are two parameters that are used to define the JBF buffer zone (See 2.4.5.3.2). The amplitude of the damping is set by $\sigma$ and the magnitude is determined by the Mach number of the freestream and whether preconditioning is used. For unpreconditioned flows $\sigma$ is set to the freestream Mach number, while unpreconditioned flows has $\sigma$ one to two magnitudes lower than the freestream Mach number. A rule of thumb used in this work is to make sure $\sigma/M$ is less than five. For these simulations the freestream Mach number is zero and $\sigma = 0.01$.

The JBF buffer zone is used in the simulation with three different values of $\beta$: two, three, and four. As the value is increased, the profile of damping increases such that the damping is concentrated closer to the boundary of the domain. The results of the three values can be compared to the exact solution in Figure 3.2. The pressure is plotted at five different nondimensional time steps as the wave propagates out of the domain. Each $\beta$ is effective in damping the solution as it travels through the buffer zone but the higher $\beta$, the less the disturbance is damped. A reflection of the acoustic wave is experienced for $\beta = 4$ due to the severe increase of damping close to the boundary. From these results a $\beta$ equal to three is recommended for use in all simulations.

![Figure 3.2. Propagation of two dimensional acoustic pulse. The colored lines are: red is $\beta = 2$, green is $\beta = 3$, blue is $\beta = 4$, and black is the exact solution.](image-url)
3.2 Preconditioner

3.2.1 Validation of Inviscid Preconditioner

To validate the inviscid preconditioner a uniform flow problem is simulated. A uniformly spaced grid that contains 201 points, spanning two nondimensional units in each coordinate direction, with the center of the domain at (0,0,0). The initial condition is a uniform mean flow in the +x direction with a random $u$ velocity fluctuation of ±20% of the mean flow imposed onto the mean flow. An example of an initial condition for a Mach number of 0.25 is provided in Figure 3.3. The Mach numbers tested range from $1 \times 10^{-4}$ to 0.5 and are conducted for three simulation methods: unpreconditioned (UNP), preconditioned with scalar artificial dissipation (SCA), and preconditioned with matrix artificial dissipation (MAT). A buffer zone is applied on the outer 0.5 of the domain with a varying $\sigma$ and $\beta = 3$. Extrapolation boundary conditions are used on all boundaries. A listing of the various parameters used for each test case is given in Table 3.1.

![Initial streamwise velocity distribution for Mach number of 0.25.](image)

Figure 3.3. Initial streamwise velocity distribution for Mach number of 0.25.
Table 3.1. Description of test cases to validate inviscid preconditioner. $d_6$ is the coefficient of artificial dissipation.

<table>
<thead>
<tr>
<th>Case Number</th>
<th>Preconditioning?</th>
<th>Artificial Dissipation</th>
<th>$\sigma$</th>
<th>Mach Number</th>
<th>$d_6$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Unpreconditioned</td>
<td>Scalar</td>
<td>$1 \times 10^{-1}$</td>
<td>0.5</td>
<td>0.005</td>
</tr>
<tr>
<td>2</td>
<td>Unpreconditioned</td>
<td>Scalar</td>
<td>$1 \times 10^{-1}$</td>
<td>0.1</td>
<td>0.001</td>
</tr>
<tr>
<td>3</td>
<td>Unpreconditioned</td>
<td>Scalar</td>
<td>$1 \times 10^{-3}$</td>
<td>0.01</td>
<td>0.001</td>
</tr>
<tr>
<td>4</td>
<td>Unpreconditioned</td>
<td>Scalar</td>
<td>$1 \times 10^{-3}$</td>
<td>0.001</td>
<td>0.001</td>
</tr>
<tr>
<td>5</td>
<td>Unpreconditioned</td>
<td>Scalar</td>
<td>$1 \times 10^{-2}$</td>
<td>0.0001</td>
<td>0.001</td>
</tr>
<tr>
<td>6</td>
<td>Preconditioned</td>
<td>Scalar</td>
<td>$1 \times 10^{-2}$</td>
<td>0.5</td>
<td>0.005</td>
</tr>
<tr>
<td>7</td>
<td>Preconditioned</td>
<td>Scalar</td>
<td>$1 \times 10^{-2}$</td>
<td>0.1</td>
<td>0.001</td>
</tr>
<tr>
<td>8</td>
<td>Preconditioned</td>
<td>Scalar</td>
<td>$1 \times 10^{-3}$</td>
<td>0.01</td>
<td>0.001</td>
</tr>
<tr>
<td>9</td>
<td>Preconditioned</td>
<td>Scalar</td>
<td>$1 \times 10^{-4}$</td>
<td>0.001</td>
<td>0.001</td>
</tr>
<tr>
<td>10</td>
<td>Preconditioned</td>
<td>Scalar</td>
<td>$1 \times 10^{-5}$</td>
<td>0.0001</td>
<td>0.001</td>
</tr>
<tr>
<td>11</td>
<td>Preconditioned</td>
<td>Matrix</td>
<td>$1 \times 10^{-2}$</td>
<td>0.5</td>
<td>0.080</td>
</tr>
<tr>
<td>12</td>
<td>Preconditioned</td>
<td>Matrix</td>
<td>$1 \times 10^{-2}$</td>
<td>0.1</td>
<td>0.016</td>
</tr>
<tr>
<td>13</td>
<td>Preconditioned</td>
<td>Matrix</td>
<td>$1 \times 10^{-3}$</td>
<td>0.01</td>
<td>0.016</td>
</tr>
<tr>
<td>14</td>
<td>Preconditioned</td>
<td>Matrix</td>
<td>$1 \times 10^{-4}$</td>
<td>0.001</td>
<td>0.016</td>
</tr>
<tr>
<td>15</td>
<td>Preconditioned</td>
<td>Matrix</td>
<td>$1 \times 10^{-5}$</td>
<td>0.0001</td>
<td>0.016</td>
</tr>
</tbody>
</table>

This problem is selected as the time required to convect the nonuniform velocity (i.e., “errors”) out of the domain is directly proportional to the Mach number of the flow. The slower the mean flow, the smaller the time step that is required for stability, and the greater the number of iterations that are required to convect the perturbations out of the domain. Thus the overall computational time increases for slower flows. In this manner the effectiveness of the preconditioner can be compared to an unpreconditioned simulation.

It should be noted that the artificial dissipation coefficient is varied between the scalar and matrix versions to keep the level of smoothing constant. Due to formulation differences between the two methods an equivalent background coefficient for matrix dissipation is sixteen times larger than a scalar coefficient [220]. The scalar preconditioning referred to in Table 3.1 is based on the sixth order accurate, sixth derivative artificial dissipation that is built into CHOPA and is typically used for computational aeroacoustic (CAA) calculations. Numerical tests conducted separately showed that the sixth order scalar artificial dissipation and a fourth order matrix artificial dissipation that emulates the scalar dissipation (by setting

93
\( V_l = V_n = 1 \) to recover a scalar artificial dissipation) showed very similar results. In this manner the impact of the additional mathematic calculations for the matrix version on the total computational time can be understood. The unpreconditioned method uses the scalar, sixth derivative method for artificial dissipation.

The first simulation is for a Mach number of 0.5, where the preconditioning is not expected to have much effect since there is not a large disparity between the convective and acoustic wave speeds. A cursory review of Figure 3.4 shows that the benefit from preconditioning is small for this moderate Mach number. In the convergence plots the abscissa is the number of iterations that are required to drop the residual of the governing equations to \( 1 \times 10^{-9} \) and the ordinate is the \( L_2 \) norm of density, which is used to determine steady state convergence. Further review of the convergence histories in Table 3.2 shows that preconditioned codes converged in a fewer number of iterations although they require a longer time to complete. The reason for this is the additional, more complex mathematical operations that are required for the preconditioner. Thus, for this Mach number preconditioning is not recommended.

![Figure 3.4](image)

**Figure 3.4.** Residual for the three CHOPA configurations for Mach number of 0.5.

As the Mach number decreases, the benefit of preconditioning becomes evident. In Figure 3.5 the convergence history for a \( M = 0.1 \) flow is shown. The number of iterations required for convergence with the preconditioned codes is approximately half of the unpreconditioned code. However, the total computational time still
exceeds the unpreconditioned code. The preconditioner is still recommended for these low Mach numbers as it has increased accuracy over the unpreconditioned version (mostly due to the calculation of the artificial dissipation). Even though the convergence history for the preconditioned matrix and scalar artificial dissipation are very similar the matrix artificial dissipation results in a more accurate solution due to the level of dissipation added to the governing equations.

At a freestream Mach number of 0.01, which is typical for underwater flows, the preconditioned codes show their effectiveness with an order of magnitude reduction in simulation times (See Figure 3.6). The unpreconditioned code exhibits a convergence issue, where the residual can not be reduced below $10^{-8}$. This is likely due to round off errors for the low Mach number. It likely represents a lower bound for accurate simulations using the unpreconditioned equations. Other more complex flows, e.g. flows over cavities, will likely have a lower convergence bound at other Mach numbers due to the complexity of the flow, but that has not been verified.

The next reduction in Mach number is to 0.001 and a plot of the residual during the simulation is shown in Figure 3.7. Here, the necessity of a preconditioned code becomes apparent as the unpreconditioned code has a constant residual. This simulation was stopped after 300000 iterations and the residual was still on the order of $10^{-2}$. For this problem the density fluctuations are small enough that

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{figure3.5.png}
\caption{Residual for the three CHOPA configurations for Mach number of 0.1.}
\end{figure}
they can not be accurately resolved in the unpreconditioned code. This illustrates a key component of the preconditioned code that uses density and pressure decomposition to accurately predict small changes in the flow field. The preconditioned codes perform extremely well. Both the matrix and scalar artificial dissipations perform equally well.

![Residual for the three CHOPA configurations for Mach number of 0.01. The unpreconditioned version converges at 38360 iterations.](image_url)

**Figure 3.6.** Residual for the three CHOPA configurations for Mach number of 0.01. The unpreconditioned version converges at 38360 iterations.

A Mach number of 0.0001 is the lowest simulated and this again highlights the deficiencies of the unpreconditioned code. The solution using the unpreconditioned code, even though it converged, is incorrect. The unpreconditioned codes uses a scalar dissipation and this results in too much artificial dissipation added to the solution. The velocity perturbations are damped before they are propagated out of the domain. The preconditioned code with scalar dissipation does not suffer from the same problem since the preconditioned code helps to equalize the eigenvalues. This reduces the artificial dissipation added to each equation, resulting in a converged solution.
Figure 3.7. Residual for the three CHOPA configurations for Mach number of 0.001.

Figure 3.8. Residual for the three CHOPA configurations for Mach number of 0.0001.
If the convergence history plots from Figure 3.5 to Figure 3.8 are reviewed it is noticed that the preconditioner enables an almost Mach number independent convergence. This is an excellent result as various researchers [114, 139] have shown that with an effective preconditioner convergence should be independent of the Mach number. There are several caveats to this, as boundary conditions and grid aspect ratio can alter this, but for this simple problem Mach number independent results are expected. For the unpreconditioned case there is an initial quick decrease in residual as the acoustic waves are quickly propagated out of the domain, which are the waves that drive the stability criterion. Thereafter, the residual slowly decreases as the convective terms are propagated out of the domain with a time step that is much smaller than is needed for propagation of the convective terms. Conversely, the preconditioned residuals have a continuously linear decrease that is reminiscent of the quick propagation of the acoustic modes in the unpreconditioned case. This is understandable as the preconditioner essentially turns all waves into pseudo-acoustic waves.

One reason that the preconditioned residuals are not truly Mach number independent is the magnitude of the damping function for the buffer zone. The preconditioner with scalar dissipation at $M = 0.001$ is simulated with two different values for the magnitude of the buffer zone damping function. The result is a slight difference in the number of iterations required for convergence. The simulation with the higher damping magnitude had a slightly smaller number of iterations required for convergence. This can be explained in that, the greater the magnitude of the damping function, the faster the perturbations are damped, and the quicker the convergence. Unfortunately, if the magnitude is pushed high enough the complex interaction of the buffer zone and the preconditioner will overwhelm the residual and cause an instability. The uniform flow problems that are run for this study use a “rule of thumb” that for preconditioned flows the damping magnitude is one order of magnitude below the freestream Mach number. This is most likely not the optimum value (i.e. largest value), and with the scalar and matrix artificial dissipation schemes adding slightly different amounts of dissipation to the solutions, small differences in convergence are noticeable. This will ultimately alter the Mach number independent convergence.

Another comparison between the scalar and matrix artificial dissipations is in computational time. An observation of the total number of iterations for either
preconditioned code (scalar or matrix) shows that they are similar. This information is shown in Table 3.2. It should be noted that the unpreconditioned solution for \( M = 0.0001 \) is too dissipative to be considered a correct solution. The total computational time is always greater for the matrix artificial dissipation, due to the more complex calculations that are required. While it is not easily apparent for this problem, but even with the greater computational time the matrix artificial dissipation is preferred because of the increased accuracy.

Initially, the matrix artificial dissipation was implemented in CHOPA because the scalar artificial dissipation is typically too dissipative for low Mach number flows. It should be noted that when the preconditioned eigenvalues are used to scale the scalar artificial dissipation it is not overly dissipative. This fact has not been discussed in the literature. So while a preconditioned code with scalar artificial dissipation will provide similar dissipation as matrix based version, numerous researchers have also proven that matrix dissipation provides much better accuracy for low Mach number flows. Thus, the matrix artificial dissipation is a very useful addition to CHOPA.

### Table 3.2. Convergence histories for uniform flow problem. Cases 1 through 5 are unpreconditioned. *The unpreconditioned problem for Mach number of 0.001 was unable to converge.

<table>
<thead>
<tr>
<th>Case Number</th>
<th>Mach Number</th>
<th>Total Number Iterations</th>
<th>Total Computational Time (s)</th>
<th>Time per Iteration (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.5</td>
<td>6624</td>
<td>1.587E4</td>
<td>2.40</td>
</tr>
<tr>
<td>2</td>
<td>0.1</td>
<td>9551</td>
<td>1.162E4</td>
<td>1.22</td>
</tr>
<tr>
<td>3</td>
<td>0.01</td>
<td>38360</td>
<td>1.192E5</td>
<td>3.11</td>
</tr>
<tr>
<td>4</td>
<td>0.001</td>
<td>300000*</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>5</td>
<td>0.0001</td>
<td>1235</td>
<td>2.926E3</td>
<td>2.37</td>
</tr>
<tr>
<td>6</td>
<td>0.5</td>
<td>5994</td>
<td>1.889E4</td>
<td>3.15</td>
</tr>
<tr>
<td>7</td>
<td>0.1</td>
<td>4592</td>
<td>2.074E4</td>
<td>4.52</td>
</tr>
<tr>
<td>8</td>
<td>0.01</td>
<td>3716</td>
<td>1.817E4</td>
<td>4.89</td>
</tr>
<tr>
<td>9</td>
<td>0.001</td>
<td>3001</td>
<td>1.360E4</td>
<td>4.53</td>
</tr>
<tr>
<td>10</td>
<td>0.0001</td>
<td>2234</td>
<td>1.105E4</td>
<td>4.95</td>
</tr>
<tr>
<td>11</td>
<td>0.5</td>
<td>6037</td>
<td>2.023E4</td>
<td>3.35</td>
</tr>
<tr>
<td>12</td>
<td>0.1</td>
<td>4616</td>
<td>2.237E4</td>
<td>4.85</td>
</tr>
<tr>
<td>13</td>
<td>0.01</td>
<td>3758</td>
<td>1.961E4</td>
<td>5.22</td>
</tr>
<tr>
<td>14</td>
<td>0.001</td>
<td>3064</td>
<td>1.461E4</td>
<td>4.77</td>
</tr>
<tr>
<td>15</td>
<td>0.0001</td>
<td>2341</td>
<td>1.219E4</td>
<td>5.21</td>
</tr>
</tbody>
</table>
Lastly, the effectiveness of the preconditioner will be examined. A reduction in time is evident once the Mach number drops below 0.01. The matrix based artificial dissipation results in an increase in accuracy over the scalar artificial dissipation. Thus, the preconditioner is recommended for any problems where the mean flow Mach number is below 0.1. The effectiveness of the preconditioner for various test problems is given in Table 3.3.
Table 3.3. Preconditioner effectiveness. N/A signifies a non-physical solution for the unpreconditioned code. A negative reduction means the preconditioned code took longer to run.

<table>
<thead>
<tr>
<th>Case Number</th>
<th>Relative to Case Number</th>
<th>Scalar / Matrix Comparison</th>
<th>% Reduction in Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6</td>
<td>Scalar</td>
<td>16%</td>
</tr>
<tr>
<td>1</td>
<td>13</td>
<td>Matrix</td>
<td>19%</td>
</tr>
<tr>
<td>2</td>
<td>7</td>
<td>Scalar</td>
<td>44%</td>
</tr>
<tr>
<td>2</td>
<td>14</td>
<td>Matrix</td>
<td>20%</td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>Scalar</td>
<td>-556%</td>
</tr>
<tr>
<td>3</td>
<td>15</td>
<td>Matrix</td>
<td>-878%</td>
</tr>
<tr>
<td>4</td>
<td>9</td>
<td>Scalar</td>
<td>*</td>
</tr>
<tr>
<td>4</td>
<td>10</td>
<td>Scalar</td>
<td>*</td>
</tr>
<tr>
<td>5</td>
<td>11</td>
<td>Scalar</td>
<td>86%</td>
</tr>
<tr>
<td>5</td>
<td>12</td>
<td>Scalar</td>
<td>87%</td>
</tr>
</tbody>
</table>

3.2.2 Gaussian Pulse Propagation

The next validation case is a time accurate simulation of a Gaussian pulse. The previous tests of the preconditioner were for steady state simulations, where the simulation is advanced with different time steps at each grid point. Also, a nonuniform grid is used to examine how the preconditioner propagates density and pressure waves. While the steady state preconditioners are primarily concerned with accelerating the convergence an unsteady solution must make sure that the cell crossing times of waves is physical. Thus, the preconditioner must equalize acoustic and convection speeds while preserving the cell crossing times.

The Gaussian pulse problem is similar to that used for the buffer zone, see Section 3.1.2. The grid has 301 points in each direction. It is centered about the origin and extends 10 units in the $x$ and $y$ directions. The grid used is shown in Figure 3.9. A buffer zone with extrapolation boundary condition is used at all boundaries. The buffer zones comprises the border that is two units thick on all edges. The buffer zone $\beta = 3$ and $\sigma = 0.01$. The simulation is run for a nondimensional time of 20 using a time step of 0.005. While the dual time-step method uses a backward implicit method and the time step is independent of the stability of the solution the time step must be small enough to ensure accuracy of
the solution.

![Figure 3.9](image.png)

Figure 3.9. Grid used for the simulation of the propagation of a Gaussian pulse. Every other grid point is omitted for clarity.

Two different Mach numbers are examined. First, a Mach number of 0.0 is used. The preconditioner will be operating at the $\epsilon_{\text{stim}}$ Mach number that is set for CHOPA since the free stream velocity is zero. Thus, it assumes a stagnation region for the entire domain and the limiter will be enforced. The second calculation will be for a uniform freestream Mach number of 0.01 in the $+x$ direction. The pressure and density are uniform with a perturbation representing the Gaussian pulse center about the origin. The pressure and density perturbations are given as,

$$p' = A \exp \left[ -\frac{\log(2) \left( (x-x_0)^2 + (y-y_0)^2 \right)}{\alpha^2} \right]$$ (3.2)

$$\rho' = A \exp \left[ -\frac{\log(2) \left( (x-x_0)^2 + (y-y_0)^2 \right)}{\alpha^2} \right]$$ (3.3)

with $x_0 = y_0 = 0$, $\alpha = 0.2$, and $A = 0.01$.

The accuracy of the solution obtained by CHOPA is compared to an exact solution. The exact solution was developed for the linearized Euler equations by Tamm and Webb [86]. This solution is acceptable as long as the amplitude of the waves is small, hence $A = 0.01$. A detailed description of the solution method is
provided in Erwin [219]. The exact solution for the pressure at any point in the
flow field is given by,
\[
p(r, \theta, t) = \int_0^\infty A(r, \theta, \omega) \omega \cos(\omega t) \, d\omega .
\] (3.4)

\[A\] is calculated via a series of Hankel transforms, which is provided in Er-\[\[\text{win}\, [219].\]

A plot of the initial condition for density along the centerline \((y = 0)\) is given
in Figure 3.10. It should be noted that when the nondimensional equation set is
examined \(\rho/\rho_0 = p/(\rho_0 \ast c_0^2)\). For a freestream Mach number of 0.0 the comparison
between the exact solution and both an unpreconditioned and preconditioned
solution is shown in Figure 3.11. Both the unpreconditioned and preconditioned
close match the exact solution. This indicates that the code is operating properly
and giving good results. Figure 3.12 has the results for a freestream Mach number
of 0.01. Again, the comparison to the exact solution for the unpreconditioned and
preconditioned solution is excellent. Care must be taken to ensure the time step
in the outer loop of the solution is small enough to ensure high accuracy. An outer
time step of 0.01 is used for these solutions. A time step of 0.05 was too coarse to
give accurate results.

A series of runs were conducted to calculate the speed up with the precondi-
tioner. The total run time is five nondimensional units. The inner loop residual
convergence criteria was set to 1E-9 with a maximum number of iterations set
to 4000. The performance improvement shown for the preconditioner is conser-
vative as some of the inner loops hit the maximum number of iterations before
the convergence criteria. For the quiescent flow case the unpreconditioned solution
took 413555 seconds while the preconditioned solution only required 18322
seconds. The preconditioned solution was a reduction of almost two orders of
magnitude (i.e. 0.044) in solution time. The simulation with a freestream Mach
number of 0.01 had an unpreconditioned solution time of 384705 seconds and the
preconditioned solution time of 18373 seconds. This is again almost two orders of
magnitude acceleration for computational time.

This test case proved two things for the preconditioner. First, the precondi-
tioner accelerated the convergence of the solution, which results in a faster simu-
lation. Secondly, the comparison between the exact and numerical solution show
that the preconditioned equations are capable of accurately, and efficiently, predicting the propagation of acoustic waves through a slow moving medium. Both of these features are necessary to predict the noise generated from hydrodynamic flow over cavities.

Figure 3.10. Initial condition for density along centerline.
Figure 3.11. Nondimensional density for the exact, unpreconditioned, and preconditioned solution at four different time steps with a freestream Mach number of 0.0.

Figure 3.12. Nondimensional density for the exact, unpreconditioned, and preconditioned solution at four different time steps with a freestream Mach number of 0.01.
3.2.3 Viscous Preconditioner

The last test case will validate the preconditioner with a viscous boundary. This problem will test the effectiveness of a preconditioner for high aspect ratio grids as well as a no-slip wall boundary. The highly complex simulation of hydrodynamic flow over a cavity will utilize all parts of the validation problem discussed in this chapter and this is a key component. The canonical problem of laminar flow over a flat plate is selected to prove the effectiveness and accuracy of the preconditioner. A laminar solution is chosen both to remove any question regarding the turbulence model as well as to allow comparison to the exact solution.

The Reynolds number of the flow is kept low enough to ensure laminar flow. In this manner the exact solution for both density and axial velocity can be compared to the prediction from CHOPA. However, as will be discussed below the low Reynolds number of the solution may be responsible for a discrepancy between CHOPA and the exact solution for a laminar boundary layer. The first comparison will be for density. The exact solution for a density distribution in a laminar boundary layer can be calculated based on the methodology of Stewartson [221]. For comparison purposes a transformed coordinate, \( \eta \), is calculated and plotted against the nondimensional streamwise velocity. The transformation is referred to as the Dorodnitsyn-Howarth transformation. The similarity coordinate is given by,

\[
\eta = \bar{Y} \sqrt{\frac{\rho_\infty U_\infty}{2\mu_\infty x}} \tag{3.5}
\]

with the local density variation used to define \( \bar{Y} \),

\[
\bar{Y} = \int_0^y \frac{\rho}{\rho_\infty} \, dy \tag{3.6}
\]

In these equations the \( \infty \) subscript refers to freestream values and the \( y \) variable in the integration is normal to the flat plate. The distance from the leading edge of the flat plate is designated by \( x \).

Next, the exact solution for the streamwise velocity distribution normal to the flat plate is given by a solution of the Blasius equations [191]. For laminar flow the two dimensional boundary layer equations (e.g. Blasius equations) can be solved
analytically to provide the exact boundary layer velocity profile at any downstream location.

A numerical simulation is performed for several different freestream Mach numbers. The nondimensional computational domain (nondimensionalized by the flat plate half length) extends from -4 to 4 in the flowstream direction and from 0 to 4 normal to the flat plate. An adiabatic wall starts at $x = 0$ and extends to $x = 2$ for a half length of one. Using symmetry boundary conditions only the upper half of the flat plate is simulated. The grid is clustered in both directions, with the clustering centered on (0, 0). Extrapolation boundary conditions are used for the North, East, and West boundaries. The JBF buffer zone, with a thickness of two units, is used in conjunction with the extrapolation boundary conditions. Lastly, there are 121 points in the streamwise direction and 109 points normal to the flat plate. A depiction of the grid and associated boundary conditions is given in Figure 3.13.

Several different simulations were conducted to compare the exact solution to the CHOPA prediction. All comparisons are made one nondimensional unit downstream of the leading edge, e.g. $x = 1$. Three different freestream Mach numbers are run: 0.4, 0.1, and 0.01. The Reynolds number of the flow based on freestream Mach number, flat plate half length, and kinematic viscosity of the fluid is kept low to ensure laminar flow. For a freestream Mach number of 0.4 the Reynolds number is 4000. The lower Mach numbers of 0.1 and 0.01 have Reynolds number of 1000 and 400, respectively. The kinematic viscosity is a constant 0.0343
\[ m^2/s, \] which is not the value for air but the perfect gas law is still maintained. The use of a constant kinematic viscosity allows the Reynolds number to remain low. The density and axial velocity predictions from CHOPA are plotted against the exact solutions for \( M = 0.4 \) in Figure 3.14 and Figure 3.15. The comparisons for both density and viscosity against the exact solution are quite good. There is a very slight overprediction for the axial velocity outside the boundary layer, which could be related to placing the boundaries too close to the flat plate.

The numerical solutions for \( M = 0.1 \) are plotted against the exact solution in Figure 3.16 and Figure 3.17. The agreement is again good but there is still an overprediction for the quantities beyond the boundary layer. At the Mach number of 0.01, Figures 3.18 and 3.19, there is a noticeable discrepancy between the numerical and exact solution.

![Figure 3.14](image)

**Figure 3.14.** Nondimensional density for the exact, unpreconditioned, and preconditioned solution at \( x/L = 1 \) with a freestream Mach number of 0.4.

Looking over the previous solutions it is evident that the comparisons become worse as the Mach number decreases. Since both the unpreconditioned and preconditioned solutions match, the difference is not due to the preconditioning. To
Figure 3.15. Nondimensional axial velocity for the exact, unpreconditioned, and preconditioned solution at $x/L = 1$ with a freestream Mach number of 0.4.

Further investigate what is causing this discrepancy new solutions were calculated without preconditioning. Several changes were made in the way the simulation was conducted. The freestream flow is at a Mach number of 0.1 with the Method of Characteristics boundary condition. Using the MOC boundary condition simulations were conducted with and without a buffer zone to see if the buffer zone was contributing to the agreement increasing with a decrease in Mach number. The computational domain was also enlarged to see if the proximity of the boundaries to the flat plate had an effect on the solution. The domain was enlarged in the North, West, and East directions by two units and an additional twenty points were added in the freestream direction and ten points normal to the flat plate. The density and velocity predictions are compared to the exact solution in Figure 3.20 and Figure 3.21. The solution without a buffer zone has a freestream velocity that is slightly less than unity. The solution that uses a buffer zone has the velocity profile approach unity, an indication that the buffer zone works. Unfortunately the comparison to the exact solution is still not perfect. There is a slight overshoot for the solution with a buffer zone. There is an overshoot as well for the solution without a buffer zone although the freestream velocity for that prediction is just below unity indicating a possible discrepancy in the nondimensionalization. The slight discrepancy between the numerical and exact solution is not expected to
Figure 3.16. Nondimensional density for the exact, unpreconditioned, and preconditioned solution at $x/L = 1$ with a freestream Mach number of 0.1.

hinder the cavity noise simulations.

Since neither the extrapolation boundary condition nor the size of the grid appear to be the reason for discrepancy with the exact solution of possibilities were investigated. An assumption for the Blasius equation to hold is that the outer flow (i.e. beyond $u/U_{\text{inf}} > 0.99$) should be inviscid flow past a flat plate. This holds for high Reynolds number flows where the boundary layer is very thin \[191\]. The solutions conducted for the validation of the viscous predictor were of relatively low Reynolds numbers (e.g. 4000, 1000, and 400) to keep the solutions laminar and eliminate any uncertainty with regard to transition of the boundary layer to turbulent or turbulence model effects. However, the low Mach numbers of the solution are likely violating the high Reynolds number assumption for the Blasius equation. The low Reynolds number of the flow has a boundary layer thickness that displaces the outer flow and results in an overshoot that has been seen by other researchers \[222\]. This overshoot will not be captured with the Blasius equation. While this is a plausible explanation for the overshoot, van Dyke’s second order
boundary layer theory does not fully corroborate this explanation [223]. For second order theory the correction to the inviscid free stream is zero. However, this is for an infinite flat plate. van Dyke’s theory shows that for a finite length flat plate the slowing down of the flow downstream of the trailing edge can cause an acceleration of the freestream, which also explains the overshoot at the edge of the boundary layer. The asymptotic theory used by van Dyke is based on Reynolds number so it is also less valid as the Reynolds number decreases. Reviewing the information and the CHOPA solutions it appears that the overshoot relative to the Blasius solution is because of the low Reynolds number of the flow. This low Reynolds number of the flow limits the applicability of the Blasius solution.

Figure 3.17. Nondimensional axial velocity for the exact, unpreconditioned, and preconditioned solution at $x/L = 1$ with a freestream Mach number of 0.1.
Figure 3.18. Nondimensional density for the exact, unpreconditioned, and preconditioned solution at $x/L = 1$ with a freestream Mach number of 0.01.

Figure 3.19. Nondimensional axial velocity for the exact, unpreconditioned, and preconditioned solution at $x/L = 1$ with a freestream Mach number of 0.01.
Figure 3.20. Nondimensional density for the unpreconditioned solution with and without a buffer zone. The solution is plotted at $x/L = 1$ and the freestream Mach number is 0.1.

Figure 3.21. Nondimensional velocity for the unpreconditioned solution with and without a buffer zone. The solution is plotted at $x/L = 1$ and the freestream Mach number is 0.1.
To understand how the preconditioner decreased the run times a residual convergence criteria of 1E-7 is used to establish a converged solution. The unpreconditioned simulation, which will have difficulty at low Mach number flows, had residuals that did not reach the criteria. Hence it was not possible to achieve a true measure of speed up but the unpreconditioned simulations were halted at 500,000 iterations. Based on those run times a decrease in run time of two orders of magnitude are evident.
Chapter 4  
Two-Dimensional Cavity Simulations

This chapter will describe the assessment of the CHOPA code against two different two-dimensional cavity experiments. The first will compare the predicted cavity frequency against experimental measurements. The other validation case will be for wall pressure fluctuations on the cavity floor. Both of these cases will show the efficiency of the two-dimensional code.

4.1 Block Experiment

In the public literature there are few experiments conducted at low Mach numbers that measure the radiated sound for simple cavity geometries [2, 74]. The lack of measurements is primarily because most low speed cavity flows are not for simple geometries (i.e. underwater vessels or automobiles). Block’s 1977 experiment has been selected for the initial simulation [2].

4.1.1 Experimental Parameters and Measurements

Block conducted a series of experiments over a range of Mach numbers and cavity length to depth ratios ($L/D$). Part of the impetus for the measurements was to understand how cavity tone mechanism changes from deep ($L/D < 1$) to shallow ($L/D > 1$) cavities and if an analytical relationship could be determined for the cavity tone frequency. The Mach numbers tested varied in 0.01 increments from 0.05 to 0.40. For the present assessment the cavity tones at $M = 0.05, 0.1, 0.15,$
0.2, 0.3, and 0.4 were used. The L/D ratio varied from 0.1 to 8 but two L/D ratios are selected for comparison purposes: 0.78 and 2.35. Both of these cavities had a depth of 5.11 cm with a length of 4 cm for L/D = 0.78 and a length of 12 cm for the L/D = 2.35.

The experiment used a wall jet to establish a turbulent boundary layer upstream of the cavity. The maximum shear layer thickness was 0.8 cm at the cavity leading edge. The average convection velocity across the cavity opening was approximately $0.6U_{\infty}$, which is typical for cavity flows. A microphone was located 2.13 m from the cavity to capture the resulting noise from the unsteady flow over the cavity.

One purpose of the experiment was to provide understanding of analytical methods to predict the frequency of cavity tones. It is known that the mechanism of cavity tones is different for deep and shallow cavities. The frequencies of deep cavities scale like a Helmholtz resonator, where the unsteady shear layer above the cavity excites a depth mode of the cavity and radiates to the far field. The directivity of the radiated noise has a monopole directivity. This is most prevalent for lower Mach numbers ($M < 0.2$). As the cavity length increases the travel time for vortices within the shear layer coincide with the travel time of the pressure waves generated when the vortices impinge on the downstream cavity edge, resulting in a radiated tone. This is a Strouhal phenomenon and the tones are not dependent on any mode of the cavity. The directivity is expected to be almost omni-directional with a slight peak in the upstream direction. East had previously developed an analytical formula to predict a cavity tone frequency for deep cavities [24]. That formulation is given as,

$$S = \frac{1}{M} \left( \frac{L}{D} \right) \frac{0.25}{1 + A \left( \frac{L}{D} \right)^B}$$  \hspace{1cm} (4.1)

where the empirical constants $A = 0.65$ and $B = 0.75$.

In a previous work [40] Block had developed an analytical expression for cavity tone frequencies for shallow cavities,

$$S = \frac{n}{\frac{1}{k_{cv}} + M \left( 1 + \frac{0.514}{L/D} \right)}.$$  \hspace{1cm} (4.2)

In this formulation $k_{cv}$ is the convection velocity and is equal to 0.6 for this experiment.
While not discussed in Block’s report, an estimate of the acoustic mode of the cavity would help quantify measured tones as being either of a Helmholtz scaling (i.e., based on cavity depth modes) or a Strouhal scaling (i.e., based on flow parameters). Tam provides an analytical method to calculate the acoustic modes of a cavity as the asymptotic limit of \( M = 0 \) is approached [35]. While the acoustic response of a cavity depends on the excitation, which is a shear layer for cavity flows, the frequency of the acoustic modes of the cavity are a function of the cavity geometry. It is when these two frequencies coincide that a feedback loop is generated and a cavity tone can be produced. In his work he correlates an arbitrary line pressure pulse to the geometry of the cavity to determine the cavity resonance frequencies. For deep cavities, where the depth mode of the cavity is likely to resonate, the frequencies of the acoustic modes are given by the zeros of the equation for \( \delta(\omega) \). Since the frequencies are complex the real part of the complex frequency is related to the cavity mode frequency and the complex part is related to the radiation damping, with higher radiation damping for higher imaginary numbers. For the \( L/D = 0.78 \) cavity resonating in the (1, 1) (depth = 1, length = 1) mode the cavity natural frequency is 1092 Hz. This is determined based on Figure 2 from Reference [35].

A reproduction of the frequency for the cavity tones measured by Block is reproduced Figure 4.1. The variation in Mach number is on the abscissa and the cavity frequency is the ordinate. The circular symbols represent the \( L/D = 0.78 \) cavity and the square symbols are for the \( L/D = 2.35 \) cavity. In addition to plotting the experimental measurements as symbols Block’s equations for shallow cavities (solid, straight lines) and deep cavities (dashed, curved lines).

### 4.1.2 Numerical Grids

Special care must be taken in developing a consistent grid within the buffer zone region. Inside the buffer zone the grid spacing in the flow stream direction should be constant and set to the grid spacing of the first interior point. The number of grid points within the buffer zone should be at least the size of the spatial stencil (i.e., seven points).
Figure 4.1. Cavity tone frequencies for varying Mach numbers. This is Figure 4 from Reference [2].

4.1.2.1 L/D=0.78 Grid

A grid was developed for the \( L/D = 0.78 \) cavity that is sufficiently resolved to capture the turbulent boundary layer as well as the developing shear layer across the cavity opening. The domain is broken into two blocks. The upper block, which spans the domain above the cavity as well as the turbulent boundary layer from the wall has 309 points in the streamwise direction (\( x \)) and 191 points normal to...
the wall ($y$) for a total of 59k points. The cavity block has 91 points in the $x$ direction and 111 points vertically (10k points). The initial spacing from the wall is 0.0001 m and increases with a hyperbolic tangential function as the distance from the wall increases. This gives a $y^+$ value between 5 ($M = 0.05$) and 36.7 ($M = 0.4$), which is sufficient for the Spalart Allmaras turbulence model with a wall function.

The baseline grid used for the $L/D = 0.78$ cavity is shown in Figure 4.2. At almost 100 locations surrounding the cavity a time history of pressure was recorded to provide the sound pressure level. The wall pressure fluctuations on the cavity walls and floor, as well upstream and downstream of the cavity. A series of points that form a semicircle around the cavity trailing edge were also recorded. The measured locations are highlighted in red in Figure 4.3. Points 1 - 12 are upstream of the cavity wall, points 13 - 34 are from the cavity leading edge to trailing edge, points 35 - 45 are on the wall downstream of the cavity, Points 46 - 62 are on a semicircle centered on the cavity trailing edge, 63-67 and 76-81 are the floor of the cavity, 68 - 75 the upstream cavity wall, and 82 - 97 the downstream cavity wall. Some specific points should be noted: point 13, the cavity leading edge; 23, the midpoint of the shear layer; 34, the cavity trailing edge; 56, on the semicircle above the midpoint of the cavity; and 59, on the semicircle above the cavity trailing edge.

![Figure 4.2. Grid used for $L/D = 0.78$ cavity.](image-url)
Figure 4.3. Locations of pressure measurements for $L/D = 0.78$ cavity.

4.1.2.2 L/D=2.35 Grid

A grid was also developed for the $L/D = 2.35$ cavity based on the same principles used for the $L/D = 0.78$ cavity. The upper block of the domain has a greater number of points compared to the other cavity grid because it is larger. The upper block has 471 points in the streamwise direction and 191 points normal to the wall for almost 90k points. The cavity block has 251 points in the flow direction and 111 points vertically for 28k points total. The initial wall spacing is again 0.0001 m with an increase normal to the wall using a hyperbolic tangent distribution for the same $y^+$ values as the $L/D = 0.78$ cavity.

Figure 4.4 shows the numerical grid used for the $L/D = 2.35$ simulations. The pressure was recorded at 89 locations surrounding the cavity as well as within the cavity. The measurement locations for this cavity are shown in Figure 4.5. Points 1 - 10 are upstream of the cavity wall, points 11 - 34 are from the cavity leading edge to trailing edge, points 35 - 45 are on the wall downstream of the cavity, Points 46 - 62 are on a semicircle centered on the cavity trailing edge, 63-64 and 71-79 are the floor of the cavity, 65-70 and 86-87 the upstream cavity wall, and 79-85 with 88-89 for the downstream cavity wall. Of particular note is point 11, the cavity leading edge; 23, the midpoint of the shear layer; 34, the cavity trailing edge; 52, the semicircle above the midpoint of the cavity; and 60, the semicircle above the
cavity trailing edge.

![Figure 4.4. Grid used for $L/D = 2.35$ cavity.](image)

### 4.1.3 Solution Parameters

This section will discuss various parameters that were used to simulate both cavities. The simulation was performed in air with a density of 1.225 kg/m$^3$. The artificial dissipation coefficient used was 0.008. To reduce the number of points required for the solution, while ensuring that the same boundary layer thickness at the cavity leading edge as the experiment, a boundary layer profile was imposed at the inlet. The simulation was conducted for a nondimensional grid, where the cavity length is used as the characteristic length. The upper domain extends from $x/L = -14$ to 18 and $y/L = 0$ to 17.

The West, North, and East boundaries for the upper block use an extrapolation boundary condition. For $L/D = 0.78$ the South boundary for the upper block was a combination of slip wall (i = 1 to 15), no-slip wall (i = 16 to 105), block communication (to the cavity block), (i = 105 to 195), and slip wall (i = 195 to 295). The $L/D = 2.35$ upper cavity block had the same topology with the block communication occupying i = 105 to 355. For both cavities the cavity block was a mixture of no-slip wall boundaries (West, South, and East) with block communication (North).
A buffer zone is combined with the extrapolation boundary conditions to ensure a stable solution. The theory of the JBF buffer zone was been presented in Section 2.4.5.3.2. The buffer zone is a damping function that suppresses the solution to a 'mean flow' that will damp any perturbations out of the solution so that the solution will converge. Based on the results from Section 3.1.2 the ramping function, $\beta$, is set to three and the damping function, $\sigma$, is 0.1. A decrease in the ramping function from 0.3 to 0.2 increased the amplitude of spurious tones. Similarly, an increase in the ramping function above 0.1 increased the amplitude of any spurious tones that are present. If the ramping function is decreased to below 0.05 the solution diverged. For the upper computational block the spatial range for the buffer zone for both cavity sizes can be found in Tables 4.1 and 4.2. There is no buffer zone on the South boundary because that is either the wall boundary or communication to the lower cavity domain. All spatial dimensions for the buffer are nondimensional.

A physical depiction of the buffer coefficient in the buffer zone is shown in 4.6 for the $L/D = 0.78$ cavity and 4.7 for the $L/D = 2.35$ cavity. The range of the buffer coefficients goes from zero (blue) to one (red).

As discussed previously, the solution inside the buffer zone is a running time average of the prediction at the buffer zone / domain interface. The solution throughout the buffer zone is constant for all points in the streamwise direction. An incoming boundary layer is imposed at interface between the buffer zone and
computational domain. A velocity profile is estimated based on a one-seventh power law and then all other variables are calculated to match that profile. Upstream of the buffer zone/computational domain interface the velocity profile is constant for all streamwise locations. This results in a boundary layer that does not change in thickness inside the buffer zone. This has been numerically tested and shown to be effective in producing the correct boundary layer at the upstream edge of the cavity.

**Table 4.1.** Buffer zone parameters for $L/D = 0.78$ cavity.

<table>
<thead>
<tr>
<th>Buffer Orientation</th>
<th>Xbuf1/L</th>
<th>Xbuf2/L</th>
<th>Ybuf1/L</th>
<th>Ybuf2/L</th>
</tr>
</thead>
<tbody>
<tr>
<td>West</td>
<td>-14.0</td>
<td>-7.0</td>
<td>0.0</td>
<td>17.0</td>
</tr>
<tr>
<td>East</td>
<td>11.0</td>
<td>18.0</td>
<td>0.0</td>
<td>17.0</td>
</tr>
<tr>
<td>North</td>
<td>-14.0</td>
<td>18.0</td>
<td>10.0</td>
<td>17.0</td>
</tr>
</tbody>
</table>

**Table 4.2.** Buffer zone parameters for $L/D = 2.35$ cavity.

<table>
<thead>
<tr>
<th>Buffer Orientation</th>
<th>Xbuf1/L</th>
<th>Xbuf2/L</th>
<th>Ybuf1/L</th>
<th>Ybuf2/L</th>
</tr>
</thead>
<tbody>
<tr>
<td>West</td>
<td>-14.0</td>
<td>-11.5</td>
<td>0.0</td>
<td>17.0</td>
</tr>
<tr>
<td>East</td>
<td>15.0</td>
<td>20.0854</td>
<td>0.0</td>
<td>17.0</td>
</tr>
<tr>
<td>North</td>
<td>-14.0</td>
<td>20.0854</td>
<td>15.0</td>
<td>17.0</td>
</tr>
</tbody>
</table>

Since the thickness of the boundary layer at the upstream cavity edge has an effect on the development of the shear layer and any resulting cavity tones, care was taken to ensure the correct thickness. The experiment had a boundary layer thickness of 0.8 cm. The distance required for a flat plate laminar boundary layer to develop to 0.8 cm thickness is 44 cm. This corresponds to $x/L = 11$ and $x/L = 3.667$ for the two cavities ($L/D = 0.78$ and $L/D = 2.35$, respectively). On the East boundary of the computational domain an extrapolation boundary condition is used to permit disturbances to exit the domain without reflection. Within this region a slip wall is imposed on the South boundary to enhance the effectiveness of the extrapolation boundary condition. As such, an incoming boundary layer that is imposed at the inlet does not develop further until the no-slip wall boundary conditions is reached.
As the boundary layer will not develop from the computational inlet until the no-slip wall is reached, an incoming boundary layer thickness is calculated based on the distance from the no-slip wall to the cavity leading edge, called $x_{\text{develop}}$. The total distance for the boundary layer to develop to a thickness that matches the experiment is $x_{\text{TBL}}$ ($x = 44$ cm). $x_{\text{scalebuf}}$ plus $x_{\text{develop}}$ needs to equal the total flat plate length, $x_{\text{TBL}}$. This leads to a distance of 16 cm for $x_{\text{scalebuf}}$. For the
The $L/D = 0.78$ cavity the start of the no-slip wall is at $x/L_{0.78} = -7$ so $xscale_{buf} = 4$. The $L/D = 2.35$ cavity has the start of the no-slip wall at $x/L_{2.35} = -2.333$ and a value for $xscale_{buf} = 1.333$. The value of $xscale_{buf}$ is used with the one-seventh power law to calculate the boundary layer thickness to impose at the computational inlet.

The nondimensional boundary layer profiles for $M = 0.1$ and $M = 0.3$ for both cavity sizes are plotted in Figures 4.8 through 4.11. The profiles are plotted at three different locations: $x/L = -5, -3$ and 0 (i.e. at cavity leading edge). As will be discussed in more detail below, unless it is specified otherwise the reference length to nondimensionalize all spatial variables for both cavity sizes will be that of the 0.78 cavity, 4 cm.

The experimental boundary layer thickness at the upstream cavity edge is 0.8 cm. When nondimensionalized by the 0.78 cavity length (4 cm) the boundary layer thickness is 0.2. As the cavity leading edge is approached ($x/L = -3$ and 0) the boundary layer starts to be affected by the shear layer and diverges from an ideal boundary layer. An approximation of the boundary layer thicknesses for the different cavities and Mach numbers is found in Table 4.3.

![Figure 4.8](image1.png)  
![Figure 4.9](image2.png)

**Figure 4.8.** Velocity profiles for $L/D = 0.78$ at Mach number 0.1.  
**Figure 4.9.** Velocity profiles for $L/D = 0.78$ at Mach number 0.3.

<table>
<thead>
<tr>
<th>Mach number</th>
<th>L/D = 0.78</th>
<th>L/D = 0.78</th>
<th>L/D = 2.35</th>
<th>L/D = 2.35</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$x/L = -3$</td>
<td>$x/L = 0$</td>
<td>$x/L = -3$</td>
<td>$x/L = 0$</td>
</tr>
<tr>
<td>0.1</td>
<td>0.34</td>
<td>0.35</td>
<td>0.34</td>
<td>0.36</td>
</tr>
<tr>
<td>0.3</td>
<td>0.29</td>
<td>0.30</td>
<td>0.28</td>
<td>0.31</td>
</tr>
</tbody>
</table>
For all simulations the Detached-Eddy simulation based on the Spalart-Allmaras turbulence model is used. As previously discussed in Section 2.5.2 the DES model is turned off to increase the stability of the preconditioned solution. This results in a laminar calculation in the shear layer regions.

The number of processors used for each cavity simulation differs because of the total number of points. For the $L/D = 0.78$ cavity there are 24 total processors used in the simulation: 18 in the upper block and 6 in the cavity block. The $L/D = 2.35$ cavity has twice as many processors: 36 for the upper block and 12 for the cavity block.

### 4.1.4 FFT Calculation Methodology

The hydrodynamic flow over the cavity generates tones that can be characterized through the spectral density of the pressure. This provides the frequency of the cavity tones, as well as the amplitude. The Fast Fourier Transform (FFT) provides the spectral information. Unless described otherwise all cavity simulations the FFT is performed on data windows of $2^{12}$ points (4096). A Hanning window is used along with a 50% overlap between each segment. The FFT calculations are performed on a pressure signal that has the time-averaged pressure magnitude removed from it. In this fashion a zero amplitude is achieved for the zeroth frequency while not affecting the spectrum.

The spectral density obtained using six different FFT parameters are shown in Figures 4.12 through 4.14. Based on these FFT calculations the most prominent
cavity tone is determined from Figure 4.14.

Figure 4.12. Spectra with varying FFT parameters.

Figure 4.13. Spectra with varying FFT parameters.

4.1.5 $L/D = 0.78$ Cavity Solution

This section will describe the numerical predictions for the $L/D = 0.78$ cavity. Unless otherwise specified all of the predictions were using a preconditioned method. The pressure spectra as well as the density and velocity results are shown. The convection velocity within the shear layer is predicted from the pressure along a line between the cavity leading and trailing edge. Lastly, the timing statistics are given so the reduction in computational time with preconditioning can be understood.
4.1.5.1 Pressure Spectra

After the initial transients have propagated out of the domain the time history of pressure is recorded. The pressure history is used with a FFT to calculate the spectral density. There are five key points that are examined in more detail: 13, the cavity leading edge; 23, the midpoint of the shear layer; 34, the cavity trailing edge; 56, on the semicircle above the midpoint of the cavity; and 59, on the semicircle above the cavity trailing edge. The spectral predictions that are plotted against Strouhal number (St) instead of frequency have a change in amplitude to provide the correct mean square pressure. As a result the amplitude of the Strouhal plots are multiplied by $U/L$.

**M = 0.05**

The nondimensional spectral density of pressure for the $M = 0.05$ case is shown in Figure 4.15. For this graphic, and others that follow, the (a) plot is for all pressure history locations and the (b) plot is for the five selected locations discussed in the previous paragraph. The experiment measured a cavity tone at $St=2.4$, which is depicted by a vertical red line. The CHOPA analysis gives tones at 0.9766, 1.904, 2.832, and 3.76. The experimental tone is between the first and second tone predicted by CHOPA.

**M = 0.1**

The next simulation is for a Mach number of 0.1. The set up for this analysis is the same as the $M = 0.05$ except for the change in Mach number. At this
slightly higher Mach number the cavity is still expected to respond in a depth mode. Block’s experiment measured a cavity tone at a $St = 1.52$. The spectral density of nondimensional pressure is shown in Figure 4.16. CHOPA predicted cavity tones at $St = 0.95, 1.90, 2.83, 3.78, \text{and} 4.76$. The tones from CHOPA are very prominent but none are related to the experimental tone. Overall the agreement with the experiment is not good.
M = 0.15
The next analysis is for a slightly higher Mach number of 0.15. All solution parameters are unchanged except for the Mach number. The cavity is still expected to respond in a depth mode, at least until the Mach number exceeds 0.2. For this flow speed the experiment recorded two Strouhal numbers: 0.94 and 1.92. Figure 4.17 shows the spectral density of nondimensional pressure. The CHOPA code predicted cavity tones at Strouhal numbers 0.944, 1.872, 2.799, 3.743, and 4.671. In this case there is extremely good agreement between the CHOPA predictions and the experiment, although CHOPA predicts additional tones. These additional numerical tones are significantly above the background but it is possible that those higher tones were lost in the background noise of the experiment.

Figure 4.17. Nondimensional spectral density for M = 0.15. (a) All pressure history locations (b) Five select pressure history locations.

M = 0.2
The next case is Mach number 0.2. This is the highest Mach number at which the cavity is expected to respond in a depth mode. At higher Mach numbers a shear layer response is expected. Block's experiment again recorded two different Strouhal numbers for the cavity tones, 0.82 and 1.63. The CHOPA simulation showed cavity tones at Strouhal numbers of 0.708, 1.404, 2.112, 2.808, 3.516, and 4.224. The CHOPA simulation had good agreement compared to the experiment for the first two numerical tones, but the tone frequency predictions were slightly lower than the experiment. The spectral density of nondimensional pressure is plotted in Figure 4.18.
Figure 4.18. Nondimensional spectral density for $M = 0.2$. (a) All pressure history locations (b) Five select pressure history locations.

$M = 0.3$

The next to last flow speed simulation is for a Mach number of 0.3. At this flow speed the cavity is expected to generate tones by responding as a Helmholtz resonator (i.e. shear layer mode). Strouhal numbers of 0.45 and 0.91 were experimentally measured for hydrodynamically generated cavity tones. A number of cavity tones were predicted numerically: $St = 0.667, 1.343, 2.002, and 2.669$. The first two tones predicted by CHOPA are higher in frequency than the experiment and the agreement is not good. The spectral density of nondimensional pressure is shown in 4.19. A more in-depth investigation for this Mach number is in the following section.

Figure 4.19. Nondimensional spectral density for $M = 0.3$. (a) All pressure history locations (b) Five select pressure history locations.
M = 0.4

The last Mach number simulated is 0.4. It is known from previous examination of the preconditioner that flow speeds greater than 0.4 do not see any benefit using a preconditioner. In fact, the additional calculations required for the preconditioner results in much longer calculation times and it is not recommended for use. This is another higher Mach number flow where the cavity is expected to respond in a shear layer mode. The experiment measured cavity tones at Strouhal numbers of 0.38, 0.75, and 1.15. The spectral density of pressure for the preconditioned simulation is shown in Figure 4.20. In those figures a large number of spurious tones can be seen along with tones that could be related to the experimental cavity tones. The Strouhal numbers that are near the experimental tones are 0.366, 0.732, and 1.01. However, it is difficult to discriminate the tones that are near the experimental cavity tones and the tones that could be considered “spurious”. For this reason the agreement between the simulation and experiment is considered questionable.

The Mach number for this solution is appropriate for an unpreconditioned solution. The artificial dissipation used for the solution is the scalar method. The spectral density of pressure for the unpreconditioned solution is shown in Figure 4.21. The agreement with the experimental tones for this solution method is good. The first three cavity tones are at 0.342, 0.689, and 1.038 compared to the experimental tones of 0.38, 0.75, and 1.15. However, as with the other numerical predictions the number of cavity tones predicted with CHOPA is greater than what was seen in the experiment. The fact that the unpreconditioned solution for M = 0.4 is much better than the preconditioned solutions lends further credence to what was discussed in Section 3.2.1. For Mach numbers of 0.4 and above the preconditioned solution takes longer than the unpreconditioned solution and the results are much poorer.
Figure 4.20. Nondimensional spectral density for $M = 0.4$. (a) All pressure history locations (b) Five select pressure history locations.

Figure 4.21. Nondimensional spectral density for $M = 0.4$. (a) All pressure history locations (b) Five select pressure history locations.
The ratio of the computational tone to experimental tone is calculated to see if a trend can be determined. If the ratio is unity then the computational tone matches the experimental tone. The ratios for each Mach number are shown in Table 4.4. For $M = 0.05$ and 0.1 there is only a single tone, which is why Tone #2 is blank. The Strouhal numbers of the tone(s) are underpredicted for all Mach numbers except for 0.15 and 0.3. The simulation almost exactly matches the experiment for $M = 0.15$. In Section 4.1.5.4 the convection velocity will be compared to the magnitude of the cavity tone (e.g. ratio over or under the experiment). The Mach number of 0.4 is unpreconditioned and only shown for comparison purposes.

Table 4.4. Ratio of computational tone frequencies to the experimental values

<table>
<thead>
<tr>
<th>Mach Number</th>
<th>Tone #1</th>
<th>Tone #2</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.05</td>
<td>0.41</td>
<td>-</td>
</tr>
<tr>
<td>0.1</td>
<td>0.63</td>
<td>-</td>
</tr>
<tr>
<td>0.15</td>
<td>1.00</td>
<td>0.98</td>
</tr>
<tr>
<td>0.2</td>
<td>0.86</td>
<td>0.86</td>
</tr>
<tr>
<td>0.3</td>
<td>1.48</td>
<td>1.47</td>
</tr>
<tr>
<td>0.4</td>
<td>0.96</td>
<td>0.97</td>
</tr>
</tbody>
</table>

4.1.5.2 Modifications to Artificial Dissipation and Grid Density For Improved Results

Since the agreement between the numerical simulation and experiment was not as close as expected, an investigation into the effect of artificial dissipation and grid density was performed. This more detailed examination was only conducted for the two lower Mach numbers: 0.05 and 0.1.

$M = 0.05$

The first simulation decreased the artificial dissipation coefficient from 0.008 to 0.002. The spectral density of nondimensional pressure is shown in Figure 4.22. The decrease in the coefficient does not result in a better comparison to the experiment.

Then, the grid density inside the cavity was refined and the results compared. The number of points in the vertical direction of the cavity was doubled (halving
the spacing between points). Only the vertical direction was altered to see if the distribution affected the propagation of the pressure. The spectral density for all points, as well as for the five key locations mentioned above are shown in Figure 4.23. The results for this simulation are almost the same as the original grid.

Next, the combination of an increase in grid density inside the cavity and a lower artificial dissipation was used for a final simulation. The Strouhal frequencies for this analysis are 1.22, 2.44, and 3.66. In Figure 4.24 the results from the

Figure 4.22. Nondimensional spectral density for $M = 0.05$ for lower artificial dissipation coefficient. (a) All pressure history locations (b) Five select pressure history locations.

Figure 4.23. Nondimensional spectral density for $M = 0.05$ with an increased vertical distribution of grid points. (a) All pressure history locations (b) Five select pressure history locations.
initial analysis at $M = 0.05$ are compared to the solutions with the new numerical parameters. There is a shift in the cavity tones compared to the original prediction. The experimental cavity tone is at a Strouhal frequency of 2.4, which matches the second tone of the simulation. The refined simulation still has a cavity tone at a frequency below the experimental cavity tone - Strouhal of 1.22. Based on this refined simulation it is recommended to increase the grid density and lower the artificial dissipation to improve the accuracy.

Converting the cavity natural frequency (1092 Hz, see Section 4.1.1) to a Strouhal number for $M = 0.05$ gives a Strouhal number of 2.54. This is close to the experimentally measured Strouhal number of 2.4 to suggest that the cavity is responding in a depth mode. For reasons that cannot be explained the numerical simulation is not capturing the correct physics and has the cavity responding with Strouhal scaling, not Helmholtz scaling. This needs to be examined in more detail in the future, it could be related to the preconditioner affecting the propagation of pressure.

![Figure 4.24. Nondimensional spectral density for $M = 0.05$ for all points (a) Original calculation parameters (b) Increased cavity grid density and reduced artificial dissipation coefficient.](image)

**M = 0.1**

The same changes in grid density and artificial dissipation coefficient were made for a higher Mach number. However, the results for this Mach number were markedly different than the lower Mach number. A decrease in the artificial dissipation
from 0.008 to 0.002 did not produce a stable solution. Several different artificial
dissipation levels between 0.008 and 0.002 were also tried but these were also
unstable. Next, the refined cavity grid was simulated with the standard 0.008
artificial dissipation coefficient. This also resulted in a divergent solution.

The refined cavity grid with the lower artificial dissipation was then used with
a Mach number of 0.1. This resulted in a stable solution, which is compared to
the standard M = 0.1 solution in Figure 4.25. The vertical red line represents
the Strouhal number from the experiment. The refined cavity grid and reduced
artificial dissipation shifts the cavity tones higher in frequency to St = 1.1, 2.25,
3.35, 4.47, and 5.59. However, none of the predicted cavity tones match the exper-
imental tone at St = 1.52.

![Figure 4.25. Nondimensional spectral density for M = 0.1 for all points (a) Original
calculation parameters (b) Increased cavity grid density and reduced artificial dissipation
coefficient.]

### 4.1.5.3 Modifications to Convergence Criteria Improved Results

An additional investigation was conducted for the Mach number 0.3 simulations
to better understand the discrepancies between the experiment and CHOPA.

First, the number of subiterations used between each outer loop time step was
investigated. In a dual-time step calculation the calculation is broken into a series
of steady state calculations, with the convergence of each steady state calculation
decided by the number of subiterations. If at any time during the steady state
calculation the residuals drop below a user-specified value (called ERRMIN) then the steady state calculation is also considered converged. For these calculations the $L_2$ norm of the residual value used for convergence is $1E^{-6}$. A maximum number of sub iterations is used for simulations in case the steady state calculation either converges extremely slowly or is an almost constant level. When the maximum number of subiterations is reached then the solution is considered "converged" and the next time step loop is initiated. Care must be taken in determining the validity of a converged solution. A reduction in the maximum number of subiterations should lead to a reduction in the total calculation time (provided that the maximum number of subiterations was controlling the convergence).

For all preconditioned calculations the maximum number of subiterations is set at 5000. This is higher than what is expected for an unpreconditioned, dual-time step scheme, where several hundred should suffice. Another solution for the $M = 0.3$ flow speed was made but the maximum number of subiterations was reduced from 5000 to 500. It can be seen in Figure 4.26 that reducing the number of subiterations does not improve the solution. The reduction in subiterations actually increases the number of spurious tones, resulting in a poorer solution.

![Figure 4.26. Nondimensional spectral density for $M = 0.3$ for selected points (a) Maximum of 500 subiterations (b) Maximum of 5000 subiterations (standard).](image)

Next, the value of ERRMIN that determines convergence is increased from $1E^{-6}$ to $1E^{-3}$. This has a similar effect as reducing the maximum number of subiterations. The $M = 0.3$ cavity problem was run for a nondimensional time of 1800. The $L_2$ residual norm of density for the two different ERRMINs is shown in
4.27. From these plots it can be seen that a reduction in ERRMIN can be used to increase the speed of the simulation without a reduction in the accuracy of the tonal prediction. For both ERRMINs the average calculation time per grid point per iteration is 0.005 seconds. However, the total simulation time for the lower ERRMIN is 2.4466 x 10⁷ seconds and the higher ERRMIN is 4.1709 x 10⁷ seconds. This is almost a 50% reduction in simulation time. It is recommended that a lower ERRMIN is at least considered for calculations.

![Figure 4.27. Comparison of residual for different ERRMINs.](image)

To understand the effect of preconditioning the same M = 0.3 cavity simulation is undertaken for 1800 nondimensional time units using the unpreconditioned solution method. This flow speed is chosen as it is the nexus between where a preconditioned solution can start to be beneficial. For higher flow speeds a preconditioned method can actually take longer due to the additional calculations. And at lower flow speeds an unpreconditioned solution may not be stable. This flow speed allows for accurate comparisons between preconditioned and unpreconditioned methods.

The solution times for different numbers of maximum iterations is displayed in Table 4.5. The preconditioned numbers are repeated from the previous paragraph. It can be seen that dropping the number of iterations to 300 results in a divergent solution. Using 500 iterations resulted in times even lower than the preconditioned method (N.B. M = 0.3 is the edge where preconditioned starts to show a benefit
in reduced calculation time). For the unpreconditioned solution the “standard” number of subiterations, 5000, has a calculation time lower than the preconditioned solution with only 500 iterations. A comparison of the pressure spectra for the two different maximum number of subiterations is shown in Figure 4.28. This shows that the solution for a maximum of 500 or 5000 subiterations is very similar.

Table 4.5. Buffer zone parameters for $L/D = 0.78$ cavity.

<table>
<thead>
<tr>
<th>Solution Method</th>
<th>Max Iterations</th>
<th>Total Solution Time (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unpreconditioned</td>
<td>300</td>
<td>N/A, Divergence</td>
</tr>
<tr>
<td>Unpreconditioned</td>
<td>500</td>
<td>2.0225E7</td>
</tr>
<tr>
<td>Unpreconditioned</td>
<td>5000</td>
<td>2.5440E7</td>
</tr>
<tr>
<td>Preconditioned</td>
<td>500</td>
<td>2.4466E7</td>
</tr>
<tr>
<td>Preconditioned</td>
<td>5000</td>
<td>4.1709E7</td>
</tr>
</tbody>
</table>

Figure 4.28. Nondimensional spectral density of pressure for $M = 0.3$ for selected points. The solution is using an unpreconditioned method. (a) Maximum of 500 subiterations (b) Maximum of 5000 subiterations (standard).

4.1.5.4 Convection Velocity

To assist in the understanding of the physical process of the cavity tones the convection speed of the large scale vortical structures are calculated from the numerical solutions. As in Reference [224] the two-point cross correlations of the pressure
are used to calculate the convection velocity. A line made up of 22 points is drawn between the leading edge and trailing edge of the cavity. The first point at \( x/L = 0 \) and last at \( x/L = 1 \) are ignored as the zero velocity at those points corrupted the cross correlation. The cross correlation function is calculated with respect to the second point of the line at \( x/L = 0.0455 \).

At each point along the line the fluctuating pressure (i.e. mean pressure removed) is used for the FFT of the pressure. The cross correlation at each point, relative to \( x/L = 0.0455 \), is calculated from the frequency spectrum of pressure. The cross correlation is normalized by the inverse FFT of the autocorrelation for each point. Similar to a FFT the normalized cross correlation must be shifted so that the function is graphically represented correctly. The time delay will have a negative part and a positive part associated with it. The cross correlation for the positive part of the time delay needs to be reversed and moved ahead of the negative part. The normalized cross correlation coefficients are typically plotted showing only the positive time delay. The cross correlation coefficients against the time delay for \( M = 0.4 \) with an unpreconditioned method are shown in Figure 4.29.

![Normalized cross correlation coefficients for pressure between cavity leading and trailing edges for \( M = 0.4 \) using an unpreconditioned method.](image)

**Figure 4.29.** Normalized cross correlation coefficients for pressure between cavity leading and trailing edges for \( M = 0.4 \) using an unpreconditioned method.

The convection velocity is calculated by determining the separation distance of the pressure signals and the time delay, \( \tau \), that corresponds to the maximum of each correlation coefficient. The time delay is defined as,
\[ \tau = \frac{t}{L_{ref}/C_{\infty}}. \] (4.3)

A Matlab routine is used to determine the maximum cross correlation coefficient for each location along the cavity shear layer. An array is developed that has the spatial separation distance from the reference point, \( x/L = 0.045 \) as well as the time delay of where the maximum cross correlation occurs. The spatial separation is plotted against the time delay for each location, resulting in a plot similar to Figure 4.30. The slope of this curve is the convection velocity. Typically the curve is linear and the convection velocity is constant from the cavity leading to trailing edge. However, it is possible that the convection velocity changes along the shear layer. A curve fit has been applied to the convection velocity curve to calculate the slope. Since the time delay at zero spatial separation must be zero this curve fit has been calculated with the origin at \((0, 0)\). The development of the vortical structures in the shear layer is periodic, which is reflected in the plots of the normalized cross correlation coefficient - they repeat with increase time delay.

In Figure 4.30 the slope is 0.1428. Since the Navier-Stokes equations are nondimensionalized by Mach number the slope must be converted to be relative to the free stream velocity, which is how it is typically presented. This is accomplished by dividing the slope by the freestream Mach number. The resulting convection velocity is then 0.36\(U_{\infty}\). Based on previous analyses conducted by Tam and Block the typical convection velocity for cavity flows is 0.4\(U_{\infty}\) to 0.6\(U_{\infty}\) [33].

Next, the normalized cross correlation coefficients for each of the flow speeds as well as the spatial distances at varying time delays are presented and analyzed. First, the preconditioned solution for \(M = 0.05\) is shown in Figure 4.31. The convection velocity is slightly lower near the leading edge but is almost constant beyond that. Based on the linear curve fit the convection velocity is 0.26\(U_{\infty}\). The cross correlation coefficients and convection velocity for \(M = 0.15\) are similar as seen in Figure 4.33. The signal is almost phase locked again with a convection velocity of 0.29\(U_{\infty}\).
Figure 4.30. Spatial separation relative to time delay for $M = 0.4$ using an unpreconditioned method.

The same data format is provided for $M = 0.2$ in Figure 4.34. The convection velocity of $0.34U_\infty$ is almost constant from the leading edge to trailing edge. The convection velocity for $M = 0.3$ is slightly higher at $0.36U_\infty$ based on the data in Figure 4.35. The results for $M = 0.4$ are markedly different from the others, as can be seen in Figure 4.37. The cross correlation coefficients have no periodicity and are a reason that the convection velocity is so low: $0.044U_\infty$. The nonperiodicity of the cross correlation coefficients led to a difficulty in determining the peak at each location that led to the jaggedness in the spatial distance relative to time delay plot.

Two of the flow speeds were chosen to examine using an unpreconditioned solution. The results for the flow speeds of $M = 0.3$ and $M = 0.4$ are shown in Figure 4.36 and Figure 4.38. The convection velocity for is $0.3U_\infty$ and $0.36U_\infty$ for $M = 0.3$ and $M = 0.4$, respectively. These values are much closer to the expected values of 0.4 to 0.6, especially when the results for $M = 0.4$ are reviewed. These results lend further credence to the poor simulation for $M = 0.4$ with the preconditioner.
Figure 4.31. Data for convection velocity of M = 0.05 solution. (a) Normalized cross correlation coefficients. (b) Spatial separation relative to time delay.

Figure 4.32. Data for convection velocity of M = 0.1 solution. (a) Normalized cross correlation coefficients. (b) Spatial separation relative to time delay.
Figure 4.33. Data for convection velocity of M = 0.15 solution. (a) Normalized cross correlation coefficients. (b) Spatial separation relative to time delay.

Figure 4.34. Data for convection velocity of M = 0.2 solution. (a) Normalized cross correlation coefficients. (b) Spatial separation relative to time delay.

All of the convection velocities are plotted against each freestream velocity in Figure 4.39 to help understand the computational results. In this figure all convection velocities are for the preconditioned solutions with the exception of M = 0.4. Because the preconditioned simulation for that Mach number gave poor results the unpreconditioned convection velocity is used to establish trends. The convection velocity is expected to be between 0.4 and 0.6 times the free stream velocity but the numerical results have convection velocities are only between 0.25 and 0.35.
To understand why the frequencies were under or over predicted relative to the experimental cavity tone (see Section 4.1.5.1) the convection velocity was examined. Based on the under/over prediction it would be expected that the convection velocity would peak for $M = 0.3$ since that was the only frequency that was over-predicted relative to the experiment. However the convection velocity is lower for $M = 0.05$ to 0.2, then increases to about $0.35U_\infty$ for $M = 0.2$ to 0.4. This does not explain why the cavity tone frequencies do not match the experiment.

Figure 4.35. Data for convection velocity of $M = 0.3$ solution. (a) Normalized cross correlation coefficients. (b) Spatial separation relative to time delay.
Figure 4.36. Data for convection velocity of M = 0.3 unpreconditioned solution. (a) Normalized cross correlation coefficients. (b) Spatial separation relative to time delay.

Figure 4.37. Data for convection velocity of M = 0.4 solution. (a) Normalized cross correlation coefficients. (b) Spatial separation relative to time delay.
Figure 4.38. Data for convection velocity of $M = 0.4$ unpreconditioned solution. (a) Normalized cross correlation coefficients. (b) Spatial separation relative to time delay.

Figure 4.39. Convection velocity at each flow speed. The convection velocity for $M = 0.4$ is for unpreconditioned.
4.1.5.5 Comparison to Equations

In this section a comparison to the equations presented by Block (see Section 4.1.1) is made. The cavity tone frequency for a deep cavity is given in Equation 4.1 and for a shallow cavity in Equation 4.2. As discussed previously the deep cavity oscillation is for primarily lower Mach numbers \((M < 0.2)\) where the cavity length to depth ratio is less than two. When this occurs the only cavity tone is that corresponding to the depth mode (e.g. Helmholtz scaling). For the shallow cavity the length to depth ratio is greater than one and the Mach number is typically above 0.2. For cavities responding in a shear layer mode (i.e. shallow cavities) the Strouhal number depends, in part, on the convection part of the shear layer, \(k_v\). Block assumed that the convection velocities is \(0.57 \times U_{inf}\). As an alternative to that the convection velocities calculated from the numerical solution can also be used (see Section 4.1.5.4).

As discussed in Section 4.1.1 the physical phenomena that govern the production of cavity tones depends on the flow Mach number. For Mach numbers above approximately 0.2 the cavity tone is generated by a feedback mechanism of the oscillating shear layer impinging on the downstream cavity edge and reinforcing the separation at the upstream leading edge. This is described by Strouhal scaling and the resulting tone has a dipole directivity. This is described by Equation 4.2. At lower Mach numbers - and especially for deeper cavities the cavity tone is generated by the cavity responding in a depth mode. The cavity tone frequency is predicted by Equation 4.1.

A comparison between the Strouhal numbers predicted for the cavity tone by CHOPA and the experiment is given in Figure 4.40. The modes of the cavity are described by “n”. Also in the figure are curves that depict the Strouhal numbers predicted for Strouhal scaling (solid line) and Helmholtz scaling (dashed line). The convection velocity used for this figure matches that used by Block, \(k_v = 0.57\). For the \(L/D = 0.78\) the transition between Strouhal scaling and Helmholtz scaling occurs around \(M = 0.2\). The red squares show the experimental cavity tones and they transition from Strouhal to Helmholtz scaling around \(M = 0.2\). The CHOPA preconditioned predictions, shown by black circles, only exhibit Strouhal scaling. At the lower Mach numbers the Helmholtz scaling is not captured by the simulations. Because of this the comparison with the experiment is much better at the higher Mach numbers. Figure 4.41 has the same experimental plots and
curves for the equations but the predictions using a higher clustered grid for the cavity and a lower coefficient for the artificial dissipation. This results in a better solution compared to the measurements.

Figure 4.42 shows the same data but this time the predictions for Strouhal and Helmholtz equations use the convection velocity calculated directly from the CHOPA simulation. The convection velocity is significantly lower than the $k_v = 0.57$ in the previous figure. The equations that describe Helmholtz scaling are unchanged but the curves that show the Strouhal scaling are decreased linearly due to the lower convection velocity. The lower convection velocity reduces all of the Strouhal scaling predictions. The new curves are too low to provide a good comparison to either the experimental measurements of CHOPA prediction. These figures show that a convection velocity of 0.57 results in much better comparisons to the experiment. The companion figure, Figure 4.43, again shows the CHOPA cavity tones when the more refined grid and lower artificial dissipation coefficient is used. The prediction is much better when the 0.57 convection velocity is used.

![Figure 4.40](image.png)

**Figure 4.40.** Comparison of cavity tones between the CHOPA simulation and the prediction from Block’s equations using a convection velocity of $0.57U_\infty$. 

150
Figure 4.41. Comparison of cavity tones between the CHOPA simulation using a higher density grid with a lower artificial dissipation coefficient and the prediction from Block’s equations using a convection velocity of $0.57U_\infty$.

4.1.5.6 Cavity Tone Conclusion

An examination of the power spectral density of the cavity tones showed that the cavity tone has a greater tonal character for points outside the shear layer. There are five points plotted for the spectral density and those that correspond to points outside of the shear layer (#56, #59) have a lower amount of turbulent energy, and have a more coherent pressure signature (i.e. more tonal).

The results for the prediction of cavity tone frequency showed that the preconditioned solution is not accurate for Mach numbers of 0.4. In addition the investigation into grid density showed that the initial grid density was too coarse. The number of points in the vertical direction within the cavity had to be doubled to obtain a better solution. Also, the original artificial dissipation coefficient needed to be decreased from 0.008 to 0.002 for better comparisons to the experiment. Thus a refined cavity grid and lower artificial dissipation is needed to capture the pressure fluctuations.
Lastly, it is not understood why the best comparisons to the experiment are for $M = 0.15$. For Mach numbers lower than 0.15 the cavity tone frequencies are under predicted relative to the experiment. Block has shown that as the Mach number is reduced, the cavity tone frequency scaling changes from a Strouhal type scaling (based on flow speed) to a Helmholtz type scaling (based on cavity resonance). However, CHOPA did not successfully predict the transition from Strouhal to Helmholtz scaling at the lower Mach numbers and continued to predict a Strouhal type scaling for cavity tones. While the grid distribution in the cavity was increased and rerun with better results it could be that an even more refined cavity grid is needed. This is an explanation of why the cavity tones are under predicted. Frequencies for $M = 0.2$ are also under predicted. It was hoped that the convection velocity for each flow speed would shed light onto why only that flow speed has over predicted cavity tone frequencies, but this turned out to not be the case.
Figure 4.43. Comparison of cavity tones between the CHOPA simulation using a higher density grid with a lower artificial dissipation coefficient and the prediction from Block's equations using the convection velocity from the CHOPA simulation.

4.1.5.7 Flow Results

In this section plots of the density and pressure for $M = 0.1$ and $M = 0.3$ are presented. The flow solutions are similar for other Mach numbers so only these two are selected. The $M = 0.1$ is shown to represent the flow for a cavity that should exhibit Helmholtz scaling while the $M = 0.3$ case is for Strouhal scaling.

For $M = 0.05$ an animation of the pressure propagation within the computational domain was generated to determine if the cavity was responding in a depth mode (e.g. Helmholtz scaling) or longitudinal mode (e.g. Strouhal scaling). At low Mach numbers the depth mode response would be expected to be the physical mechanism, with a resulting monopole directivity. The animation did not show a depth mode for the pressure.
M = 0.1

The flow solution at a Mach number of 0.1 is discussed in this section. The solution at a nondimensional time step of 500 is chosen to represent the unsteady solution. The density is plotted in Figure 4.44. Note that this is only a portion of the entire computational domain close to the cavity. The turbulent flow within the cavity is evident in this graphic. The incoming boundary layer can be seen is evident with the shear layer undulating above the cavity. There are three distinct vortices evident with one just forming as the shear layer leaves the upstream cavity edge, one fully developed in the shear layer, and one impinging on the downstream cavity edge. Once the vortex has impinged on the downstream edge it is split and partially moves downstream along the wall and partially falls into the cavity where a recirculating flow is generated. The pressure at the same time step is shown in Figure 4.45. The three vortices are also evident in this flow solution as well. The graphics shown for this time step are a representative snapshot in time of the unsteady physical process.

Figure 4.44. Nondimensional density at \( t = 500 \).

M = 0.3

The pressure and density solutions for a Mach number of 0.3 are provided in this section. The solutions shown below are for a nondimensional time step of 500. First, the density is shown in Figure 4.46. The flow is similar to the lower Mach
number, with an incoming boundary layer and an oscillating shear layer. There are two distinct vortices in this solution. One almost in the middle of the cavity and one about to impinge on the downstream edge. There is a recirculating zone inside the cavity that is a result of the shear layer. The full domain, including the buffer zone, is shown in Figure 4.47. This graphic shows the noise emanating from the cavity and has an upstream propagation of the wave. This is typical of cavity flows [12]. The solution for pressure at the same time step is shown in Figure 4.48. The radiating pressure field can also be seen in this figure.
Figure 4.46. Nondimensional density at $t = 500$.

Figure 4.47. Nondimensional density at $t = 500$. 
Figure 4.48. Nondimensional pressure at $t = 500$. 

Pressure

- 0.7225
- 0.7175
- 0.7126
4.1.5.8 Timing

There are two key points to the preconditioned solution. First, for lower Mach number flows a preconditioned solution is required to obtain a stable solution. Second, the disparate acoustic wave and turbulence convection velocities result in very long convergence times. The preconditioner equalizes the two speeds and reduces the total simulation time.

All simulations discussed in this thesis were conducted on The Pennsylvania State Aerospace Engineering’s high-performance computing cluster (COCOA5). Each computational node in the cluster is built on the DL165 platform and uses two ADM 6276 “Interlagos” 16-core processors operating at 2.3 GHz. A total of 1,504 cores are available with this system.

To understand the effectiveness of the preconditioner the simulation times for each flow speed are examined. First, a detailed examination of the M = 0.05 simulation is shown in Figure 4.49. The timings of four different simulations are shown: original simulation parameters, lower artificial dissipation coefficient, increased grid density for cavity, and lower artificial dissipation with increased grid density. The total timings for all four simulations are comparable, with the lower artificial dissipation and improved grid taking the longest time, as expected. For all simulations the preconditioning calculations took about 25% of the total simulation time. From this it is evident that the improved solution provided by a lower artificial dissipation coefficient and increased grid density does not significantly increase the calculation time and is highly recommended for future simulations.

Next, the calculation times for various M = 0.3 simulations are given in Figure 4.50. The preconditioned time for the 5000 inner iterations is the case with the original parameter run and the timing is comparable to the M = 0.05 case. At the M = 0.3 the timings for both preconditioned solution methods were greater than the unpreconditioned solution. It had previously been discussed that a Mach number of 0.4 and above the preconditioned solution gave poor results and was not recommended. Additionally higher flow speeds had increased calculation times for the preconditioned method due to the added calculations for the preconditioner (e.g. matrix multiplication of the residual, etc).

The time to run the cavity flow simulation for 1800 nondimensional time units is shown in Figure 4.51. The total calculation time is in red and the portion of the calculation time for the preconditioner is in blue. There are several interesting
Figure 4.49. Simulation timings for $M = 0.05$ flow speed.

Figure 4.50. Simulation timings for $M = 0.3$ flow speed.

points to be made from this graph. The preconditioned calculation time for the $M = 0.3$ and 0.4 flow speeds is longer than the unpreconditioned time, as was previously discussed. The reason for that lies in an examination of the subiterations needed for convergence. The number of subiterations for convergence at each time step was much higher for the preconditioned simulation, hence the higher calculation time.
The total calculation time decreases linearly from the $M = 0.3$ solution until the $M = 0.05$ solution is examined. The total calculation time for $M = 0.05$ is, again, based on the number of subiterations at each time step. There are approximately 900 subiterations at each time step, significantly higher than other flow speeds. It is unknown why the number of subiterations are so much higher for just this flow speed.

![Figure 4.51. Simulation time for each flow speed.](image)

**4.1.5.9 $L/D = 0.78$ Conclusions**

The power spectral densities predicted by CHOPA using a preconditioned method did not match the experiment for a Mach number of 0.4. However, the unpreconditioned solution was able to accurately predict the cavity tones at $M = 0.4$. The preconditioner has already been shown to be accurate at a range of Mach numbers so it is unknown why this is occurring in the present case. It could be related to the matrix-based artificial dissipation used for the preconditioner. At lower Mach numbers it is scaled appropriately to introduce a lower level of artificial dissipation to provide a more accurate solution but it may not be providing enough dissipation at the higher Mach numbers, leading to a marginally unstable solution. An additional investigation into the 0.05 and 0.1 solutions showed that a lower artificial dissipation coefficient and greater grid density is required.
The cavity tones for lower speeds, $M = 0.3$ and below, matched the experimental tones well with the best prediction at $M = 0.15$. At speeds below $M = 0.15$ the cavity tones predicted by CHOPA did not transition from a Strouhal type of scaling (based on flow speed) to a Helmholtz type scaling (where the cavity resonance frequency dominates). This could be an indication of insufficient grid resolution, even though the cavity solution on a grid with higher resolution was examined in Sections 4.1.5.2 and 4.1.5.3 with better results.

At almost all flow speeds, there were a number of additional tones that were predicted at frequencies greater than the experimental measurements. When plotted on a decibel scale the higher tones appear prominent but in reality these tones are often quite lower than the experimentally measured tones. These could have existed in the experiment but gone unmeasured due to higher ambient noise. For the highest flow speed, $M = 0.4$ there were so many additional tones that it was difficult to determine which was related to a cavity flow tone and which were spurious.
4.1.6  \( L/D = 2.35 \) Cavity Solution

Cavities with a \( L/D \) range from one to eight are classified as shallow, open cavities where the shear layer spans the entire cavity [30]. The shear layer is responsible for the generation of the cavity tones with this configuration. Unless otherwise stated the predictions for the \( L/D = 2.35 \) cavity in this section are for a solution using a preconditioned method. Pressure spectra, density, and velocity results will be shown. As with the \( L/D = 2.35 \) cavity the convection velocity along the shear layer is predicted from the pressures. The differences in calculation times for preconditioned and unpreconditioned solutions are shown to demonstrate the repercussions of using a preconditioned method.

The numerical solution is performed on a nondimensional grid as the solution is calculated with the nondimensional RANS equations. A nondimensional solution for the \( L/D = 2.35 \) cavity using the physical cavity length of twelve cm did not result in a stable solution. The reasons for the instability are unknown. A stable solution was achieved with exactly the same grid when it was nondimensionalized by a reference length of four cm. Four cm is the cavity length for the \( L/D = 0.78 \) cavity. This results in a nondimensional cavity length of three for the \( L/D = 2.35 \) simulations.

4.1.6.1 Pressure Spectra

The time history of eighty-nine different sampling locations were recorded. This pressure history is used in a FFT to calculate the spectral density function of pressure. Out of the sampling points five points were again marked for closer inspection: 11, the cavity leading edge; 23, the midpoint of the shear layer; 34, the cavity trailing edge; 52, on a semicircle above the midpoint of the cavity; and 60, on a semicircle above the cavity trailing edge. The spectral density of pressure is plotted against Strouhal.

\( M = 0.05 \)

At this flow speed there are no cavity tones predicted with CHOPA nor were there tones measured during the experiment. Typically, at low Mach numbers cavity tones are generated from a depth mode response (i.e. Helmholtz scaling). The depth mode is more difficult to excite for shallow cavities. The nondimensional
spectral density of pressure for $M = 0.05$ is shown in Figure 4.52. In this graphic, and with all others that follow, the (a) plot is for all pressure locations and the (b) plot is for the five selected locations of interest that were discussed previously. When cavity tones are present the Strouhal frequency at which they occur will be highlighted by a vertical red line.

![Figure 4.52](image)

**Figure 4.52.** Nondimensional spectral density for $M = 0.05$. (a) All pressure history locations (b) Five select pressure history locations.

**M = 0.1**

Next is the Mach number 0.1 simulation. At this still relatively low Mach number there were two cavity tones recorded in the experiment at Strouhal numbers of 1.70 and 2.28. The predicted spectral density of nondimensional pressure is shown in Figure 4.53 where a cavity tone at 0.18 and 0.29 is shown. The predicted cavity tones are well below experiment. It is unknown why there is an order of magnitude difference between the experimental and computational cavity tones.

**M = 0.15**

The analysis for this section is for the next higher Mach number: $M = 0.15$. At this flow speed the experiment yielded cavity tones at three Strouhal numbers: 1.70, 2.30, and 2.81. The spectral density of nondimensional pressure, with red lines defining the experimental tones, is shown in Figure 4.54. The cavity tones predicted by CHOPA are for Strouhal numbers 0.18 and 0.28. The predicted tones are again at much lower frequencies than the experimental tones. It is unknown why there is a discrepancy between the measurement and simulation.

163
M = 0.2
The next simulation is for a Mach number of 0.2. Block’s experiment recorded cavity tones at Strouhal numbers of 1.05, 1.63, 2.12, and 2.44. The CHOPA simulation had cavity tones at 0.16, 0.34, 0.51, and 0.65. The agreement for the CHOPA-predicted cavity tones against the experiment is poor. The spectral density of nondimensional pressure is plotted in Figure 4.55.

Figure 4.53. Nondimensional spectral density for M = 0.1. (a) All pressure history locations (b) Five select pressure history locations.
Figure 4.54. Nondimensional spectral density for $M = 0.15$. (a) All pressure history locations (b) Five select pressure history locations.

Figure 4.55. Nondimensional spectral density for $M = 0.2$. (a) All pressure history locations (b) Five select pressure history locations.
\textbf{M = 0.3}

The next flow speed simulation is for a Mach number of 0.3. The experiment measured cavity tones at Strouhal numbers of 1.02, 1.52 and 2.05 while the CHOPA simulation had cavity tones at 0.15, 0.32, and 0.50. The spectral density of nondimensional pressure is given in Figure 4.56. Again, the CHOPA-predicted cavity tones underpredict the experiment by a significant amount.

This Mach number is high enough that an unpreconditioned solution is expected to give a reasonable answer. The spectral density of nondimensional pressure is shown in 4.57. For this simulation cavity tones are predicted at Strouhal numbers of 0.16, 0.33, 0.488, 0.62, and 0.814. Again, these cavity tones are below the experimentally measured tones. The frequency of these tones are very close to the preconditioned solution. This is an indication that the discrepancy between the experimental and numerical cavity tones is not due to preconditioning. The discrepancy is unclear.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure456.png}
\caption{Nondimensional spectral density for M = 0.3. (a) All pressure history locations (b) Five select pressure history locations.}
\end{figure}

\textbf{M = 0.4}

The final simulation is for a Mach number of 0.4. As has been discussed previously the preconditioner is typically not beneficial for flow Mach numbers above M = 0.4. Instead the preconditioned solution will have a longer simulation time due to the extra calculations required for the preconditioner. The measured cavity tones were at Strouhal numbers of 0.96, 1.46, and 1.92. The preconditioned solution was
Figure 4.57. Nondimensional spectral density for $M = 0.3$ for unpreconditioned solution. (a) All pressure history locations (b) Five select pressure history locations.

not stable for this condition. The cavity tones for the unpreconditioned are shown in Figure 4.58. The predicted frequencies are again all well below the measured frequencies, leading to poor agreement with the experiment. As was seen for the $L/D = 0.78$ cavity predictions the number of cavity tones predicted with CHOPA is greater than was noted in the experiment.

Figure 4.58. Nondimensional spectral density for $M = 0.4$. (a) All pressure history locations (b) Five select pressure history locations.
4.1.6.2 Convection Velocity

The convection speed of the large scale vortical structures can be used to understand the development of cavity tones. As was done for the $L/D = 0.78$ cavity simulations two-point cross correlations of the pressure are used to determine the convection velocity. A line between the leading edge and trailing edge of the cavity is generated from twenty four individual points. The cross correlation function is calculated with respect to the second point on the line at $x/L = 0.17303$. The first and last points were omitted since the zero velocity at this location results in a zero cross-correlation. The details of the calculation of the convection velocity were given in Section 4.1.5.4. The typical convection velocity for cavity flows is expected to be $0.4U_{\infty}$ to $0.6U_{\infty}$ [33].

As was done for the $L/D = 0.78$ cavity the normalized cross correlation coefficients for each flow speed are plotted relative to the time delay. The peaks in the cross correlation coefficients are used to plot spatial separation distances against the corresponding time delay. The separation distance varies from zero (cavity leading edge) to one (cavity trailing edge). The slope of the curve is the convection velocity. A linear curve fit to the data is used to estimate the slope.

For each of the simulations previously discussed the spatial distances at varying time delays are presented in Figure 4.59 through Figure 4.65. These are the plots used to understand the convection velocity as you move from the leading edge to trailing edge of the cavity. The convection velocity for each of the simulations is listed in Table 4.6. It is immediately seen that the convection velocities for all of the simulations for the shallow cavity are lower than expected. For shallow cavities the cavity tones are due to a feedback loop that is set up with the shear layer impinging on the cavity trailing edge and traveling upstream to the separation point at the leading edge. There the acoustic wave excites instability waves in the shear layer, completing the feedback loop. Block has theorized that the Strouhal number of these shallow cavities can be determined with Equation 4.2. For the CHOPA simulations the lower convection velocity $(k_v)$ dominates the Strouhal number. The lower convection velocities coupled with this equation show why the Strouhal numbers are lower than the experiment. However, it is unknown why the convection velocities are lower than anticipated. The next section will review the flow solutions to see if anything can be gleaned from the unsteady flow.
Figure 4.59. Spatial separation relative to time delay for $M = 0.05$ solution.

Figure 4.60. Spatial separation relative to time delay for $M = 0.10$ solution.
Figure 4.61. Spatial separation relative to time delay for $M = 0.15$ solution.

Figure 4.62. Spatial separation relative to time delay for $M = 0.20$ solution.
Figure 4.63. Spatial separation relative to time delay for M = 0.30 solution.

Figure 4.64. Spatial separation relative to time delay for M = 0.30, unpreconditioned, solution.
Figure 4.65. Spatial separation relative to time delay for $M = 0.40$ solution.

Table 4.6. Nondimensional boundary layer thickness.

<table>
<thead>
<tr>
<th>Mach number</th>
<th>Convection Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.05</td>
<td>$0.1236U_\infty$</td>
</tr>
<tr>
<td>0.10</td>
<td>$0.0752U_\infty$</td>
</tr>
<tr>
<td>0.15</td>
<td>$0.1997U_\infty$</td>
</tr>
<tr>
<td>0.20</td>
<td>$0.1339U_\infty$</td>
</tr>
<tr>
<td>0.30</td>
<td>$0.1242U_\infty$</td>
</tr>
<tr>
<td>0.30 UNPREC</td>
<td>$0.062u_\infty$</td>
</tr>
<tr>
<td>0.40</td>
<td>NAN</td>
</tr>
<tr>
<td>0.40 UNPREC</td>
<td>$0.1552U_\infty$</td>
</tr>
</tbody>
</table>
4.1.6.3 Cavity Tone Conclusion

Reviewing the power spectral densities for all of the different flow speeds reveals several things. First, unlike the \( L/D = 0.78 \) deep cavity the sharpness of the cavity tone is independent of location of the measuring point. Previously points outside the shear layer were sharper than other points. However, the amplitude of pressure does vary depending upon position, with the highest amplitude being that directly above the cavity trailing edge. The shallow cavity tone is controlled by a feedback loop that is initiated by an acoustic wave generated when the shear layer impinges on the cavity trailing edge. Thus, the sound is generated at that point, which leads to the greatest amplitude.

As occurred with the \( L/D = 0.78 \) cavity simulations the preconditioned solution for a Mach number of 0.4 was not accurate. In fact, for the \( L/D = 2.35 \) cavity the solution was not stable and could not be completed. Lastly, it is not understood why the predicted cavity tone frequencies are all significantly lower than the experimentally measured cavity tones. The significantly lower than expected convection velocity in the shear layer directly translates to a lower cavity tone frequency. However, it is not understood why the convection velocities are lower. The next section will examine the flow field results in the hope of shedding more light on the cavity tone frequency discrepancy.

4.1.6.4 Flow Results

This section will give plots of the density and pressure for the \( M = 0.1 \) and \( M = 0.3 \) flow speeds. The flow solutions for other speeds are similar to these two so only these are shown for brevity.

\( M = 0.1 \)

This section reviews the fluid dynamic solution for a Mach number of 0.1. The simulation was conducted for over four thousand nondimensional time units. Results are shown for the computational domain that does not include the buffer zone regions. The solution within the buffer zone is non-physical, but mathematically correct, so it will be excluded in the review. The density at this time step is plotted in Figure 4.66. The development of vortices in the shear layer is evident in this plot. The zoomed plot of the density for the same time step is shown as well.
There are two distinct vortices in this graphic as well as a third that is impinging on the cavity trailing edge. The pressure at the same time step is shown in Figure 4.67. The positive and negative regions of pressure that represent vortices are easily seen. Other time steps are represented by this snapshot - the vortices are convected downstream with the flow. The extrapolation boundary condition with a buffer zone is shown to be working very effectively. There are no pressure reflections at the start of the buffer zone nor at the edge of the computational domain.
Figure 4.66. Nondimensional density at $t = 3000$ for $M = 0.1$. (a) Full computational domain. (b) Zoom around cavity.

Figure 4.67. Nondimensional pressure at $t = 3000$ for $M = 0.1$. (a) Full computational domain. (b) Zoom around cavity.

M = 0.3
The next higher flow speed of Mach number $= 0.3$ is reviewed in this section. The solutions are again shown at a nondimensional time step of $3000$. A plot of the density in the computational domain is shown in 4.68. The flow is similar to the lower Mach number with distinct vortices in the cavity shear layer and a pressure wave that emanates from the cavity. The flow is damped as the edge of the computational domain is approached due to the buffer zone. Reviewing the
close up of the solution of just the cavity shows how the vortex is moving in the cavity shear layer. The pressure field at the same time step is shown in Figure 4.69. A sound wave that is propagating away from the cavity, with a slight upstream directivity to it, can be seen in that figure.

![Figure 4.68](image1)

(a) (b)

**Figure 4.68.** Nondimensional density at $t = 3000$ for $M = 0.3$. (a) Full computational domain. (b) Zoom around cavity.

![Figure 4.69](image2)

(a) (b)

**Figure 4.69.** Nondimensional pressure at $t = 3000$ for $M = 0.3$. (a) Full computational domain. (b) Zoom around cavity.

### 4.1.6.5 Timing

In addition to allowing for a stable solution for low Mach number flows the preconditioner equalizes the acoustic and turbulence convection velocities to reduce the
total simulation time. In this section the simulation times for each flow speed will be discussed.

The total time to run each cavity flow simulation for 2000 nondimensional time units is given in Figure 4.70. The total time includes the time for all processors, which in this case was forty-eight (thirty-six processors for the upper block and twelve processors for the lower, i.e. cavity, block). This is significantly greater than the wall clock time (forty-eight times greater).

For the timing figure the total calculation time is in red and the portion of the calculation time that represents the calculations necessary for the preconditioner is in blue. The total calculation time for all flow speeds except for $M = 0.05$ is similar. This is expected when using a preconditioned solution as the preconditioner equalizes the turbulence convection and acoustic wave speeds for any Mach number, which results in similar calculation times. The $M = 0.05$ simulation time is different than the other flow speeds as there are a larger number of subiterations required at each time step for convergence. The larger number of subiterations requires more calculation time overall. The preconditioner has been shown to be effective at these lower Mach numbers so it is unknown why the calculation time significantly increases for $M = 0.05$. More time is spent with the preconditioner calculations as the number of subiterations increases. The relatively constant calculation times for the flow speeds is an indication that the preconditioner works well across the flow speeds investigated.
Figure 4.70. Simulation timings for each flow speed with $L/D = 2.35$ cavity.

4.1.6.6 $L/D = 2.35$ Conclusions

A review of all of the predicted cavity tones for each flow speed reveals a common thread through all of the simulations. Except for the $M = 0.05$ prediction, which had no cavity tone measured or predicted, all of the CHO P A predictions significantly underpredicted the frequency of the cavity tone. See Table 4.7 for a list of cavity tone frequencies. Additionally, all of the cavity tones were similar in frequency between each flow speed. These trends were consistent regardless of whether a preconditioned or unpreconditioned solution method. It is possible that the cavity tones seen in the CHOPA predictions were not due to directly radiated noise from an impinging shear layer on the cavity trailing edge but from standing waves due to the grid and numerical solution. However, a review of the flow solution did not reveal any standing waves or any phenomena other than the expected cavity flow.
Table 4.7. $L/D = 2.35$ convection velocities and cavity tone Strouhal numbers for all flow speeds.

<table>
<thead>
<tr>
<th>Mach number</th>
<th>Convection Velocity</th>
<th>Velocity</th>
<th>Strouhal Numbers</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.05</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>0.10</td>
<td>0.18</td>
<td>0.29</td>
<td>-</td>
</tr>
<tr>
<td>0.15</td>
<td>0.18</td>
<td>0.28</td>
<td>-</td>
</tr>
<tr>
<td>0.20</td>
<td>0.16</td>
<td>0.34</td>
<td>0.51</td>
</tr>
<tr>
<td>0.30</td>
<td>0.15</td>
<td>0.32</td>
<td>0.50</td>
</tr>
<tr>
<td>0.30 UNPREC</td>
<td>0.16</td>
<td>0.33</td>
<td>0.49</td>
</tr>
<tr>
<td>0.40</td>
<td>NAN</td>
<td>NAN</td>
<td>NAN</td>
</tr>
<tr>
<td>0.40 UNPREC</td>
<td>0.15</td>
<td>0.32</td>
<td>0.46</td>
</tr>
</tbody>
</table>
4.2 Stallings Experiment

The two previous sections tested ability for the preconditioned CHOP A code to accurately predict cavity tone frequencies. This assessed the CAA tool’s ability to predict coupling between the hydrodynamic flow and the resulting pressure field. These sections examined the physical mechanisms behind the radiation from a deep cavity ($L/D < 1$) where the shear layer excites a depth mode of the cavity to generate a tone and from a shallow cavity ($L/D > 1$) where a feedback loop between the cavity trailing and leading edge generates a tone that directly radiates to the far field. In this section the work of Stallings tal [3] is used to make comparisons for the wall pressures on the cavity. The cavity wall pressures have significant consequences, since for low Mach number flows, and especially for under water flows, the wall pressure fluctuations can drive a structural resonance of the cavity walls that can lead to structural fatigue or significant amplitude cavity tones.

4.2.1 Experimental Parameters and Measurements

Stallings conducted a large number of experiments with a variety of subsonic Mach numbers (0.2 - 0.95) for two different sized cavities: $L/D = 5.42$ and $L/D = 6.25$. These are both shallow cavities based on their geometry and any cavity tones that would be generated would be due to a shear-layer feedback between the cavity leading and trailing edge. The focus of Stallings tal’s work was to understand the effect of shear layers on stores inside a cavity but there were experiments conducted without any stores in the cavity that provide useful validation data. The wall pressure fluctuations were recorded by Stallings tal at multiple points on the cavity floor. The time average of these fluctuations was used to assess the CHOP A predictions. While a range of Mach numbers were completed by Stallings it has been shown that a preconditioned solution is typically beneficial for Mach numbers below 0.4. Since the code has been validated previously for moderate subsonic flow speeds [12] only, low Mach number flows are considered for this section. Numerical simulations at a Mach number of 0.2 are presented below. An additional solution for the $L/D = 5.42$ cavity at a Mach number of 0.1 is also discussed to provide further insight into the prediction of cavity flows.
4.2.2 Numerical Grids

The information gained in developing the grids for the Block simulations is used to generate an accurate grid for the Stallings simulations. The grid spacing within the buffer zone is constant in the flow direction. Sufficient resolution is used to capture the turbulent boundary layer upstream of the cavity and the shear layer on the cavity opening.

4.2.2.1 L/D=5.42 Grid

The computational domain is separated into two blocks for the solution. The upper block has 613 points in the $x$ direction and 235 points normal to the wall in the $y$ direction. The lower, or cavity block, has 151 points in the flow direction and 201 points normal to the cavity floor. This results in 144055 points in the upper block and 30,351 points in the lower block. The initial wall spacing is 0.00015 m for a $y^+$ value of 15 for the M = 0.2 case, which is acceptable for the Spalart-Allmaras turbulence model with a wall function. A view of the grid is given in Figure 4.71.

![Figure 4.71. Grid for L/D = 5.42 cavity simulation. (a) Full grid. (b) Zoom around cavity block](image)

4.2.2.2 L/D=6.25 Grid

The grid for the $L/D = 6.25$ grid is very similar to the $L/D = 5.42$ case, as can be seen from Figure 4.72. Each of the blocks have the same number of points as the other cavity.
4.2.3 Solution Parameters

This section provides the different parameters that are set to develop the cavity solutions. The simulation was conducted with air as the fluid medium at a density of 1.225 kg/m$^3$. The matrix artificial dissipation coefficient is 0.008. To reduce the number of grid points in the solution, an artificial boundary layer with a thickness of 0.45” (11.43 mm) was imposed at the inlet of the computational domain. The grid is nondimensionalized by the cavity length. The upper domains extend from $x/L = -14$ to $x/L = 18$ and $y/L = 0$ to $y/L = 17$.

The West, North, and East boundaries of the upper block of the computational domain use an extrapolation boundary condition. On the South boundary of the $L/D = 5.42$ cavity a mixture of slip-wall ($i = 1$ to 113), no-slip wall ($i = 114$ to 225), block communication ($i = 225$ to 375), and then no-slip and slip-wall to the end of the domain. The $L/D = 6.25$ cavity has the same topology. The cavity block uses no-slip wall boundary conditions on the West, South, and East boundaries with a block communication boundary on the North side.

The extrapolation boundary condition is used in conjunction with a buffer zone to ensure all flow structures and acoustic waves exit the domain smoothly without reflection. Based on the numerical simulations conducted previously and the solutions of the Block cavities, the $\beta$ ramping function is set to three and the damping function $\sigma$ is set to 0.1. The buffer zone’s extent is given in Table 4.8. The South boundary does not have a buffer zone as it is either a wall or a block.
communication. The buffer zone contours for the $L/D = 5.42$ cavity are shown in Figure 4.73. The buffer zone coefficients range from zero (blue) to one (red). Areas where the buffer zone coefficient are zero means the computational solution is unaffected by the buffer zone. The solution within the buffer zone is a running time average of the solution at the buffer zone / domain interface. This solution is constant for all points along a line in the streamwise direction.

**Table 4.8.** Buffer zone parameters for $L/D = 5.42$ and $L/D = 6.25$ cavities.

<table>
<thead>
<tr>
<th>Buffer Orientation</th>
<th>Xbuf1/L</th>
<th>Xbuf2/L</th>
<th>Ybuf1/L</th>
<th>Ybuf2/L</th>
</tr>
</thead>
<tbody>
<tr>
<td>West</td>
<td>-14.0</td>
<td>-8.0</td>
<td>0.0</td>
<td>17.0</td>
</tr>
<tr>
<td>East</td>
<td>13.0</td>
<td>18.0</td>
<td>0.0</td>
<td>17.0</td>
</tr>
<tr>
<td>North</td>
<td>-14.0</td>
<td>18.0</td>
<td>14.0</td>
<td>17.0</td>
</tr>
</tbody>
</table>

For the numerical model to accurately reflect the experiment, the boundary layer approaching the cavity must match the experimental boundary layer. The boundary layer thickness at the cavity leading edge is 0.45 inches. This translates to a $\delta/L$ of 0.0173 for the $L/D = 5.42$ cavity and 0.0150 for the $L/D = 6.25$ cavity. Since the boundary layer thickness is relatively small the boundary layer naturally develops along the no-slip wall just upstream of the cavity. If it were thicker then the same methodology as laid out in Section 4.1.3 could be used to ensure a proper boundary layer at the cavity leading edge.
As with all cavity simulations in this study the Detached-Eddy simulation, based on the Spalart-Allmaras turbulence model, is used. The DES model is again turned off to improve the stability of the preconditioned solution. When the DES model is turned off it results in a laminar calculation in the shear layer regions close to the wall.

Lastly, the number of processors used for the cavity simulations is selected to evenly distribute the points with almost the same number of points for each processor. Processors that have more points than others will require longer time to complete their calculations and results in lag time for the processors that have a smaller number of grid points. This can result in inefficiencies. For the $L/D = 5.42$ cavity the upper block had 54 processors and the lower cavity block had 12 processors. Since the number of points in the computational domain of each cavity is similar, the $L/D = 6.25$ cavity uses the same distribution of processors.

### 4.2.4 $L/D=6.25$ Cavity Solution

In this section the numerical results of the $L/D = 6.25$ at a Mach number of 0.2 are shown. The time-averaged wall pressure fluctuations for both a preconditioned and unpreconditioned solution will be shown as well as visualizations of the flow.

#### 4.2.4.1 Flow Results

The simulation is conducted long enough to have removed all of the initial transients from the flow. Once the flow reached a quasi-steady state, where the shear layer has developed, then a time averaging of the wall pressure fluctuations were recorded. Both an unpreconditioned and preconditioned solution were completed for this cavity $L/D$. Since it is of more interest, the flow for the preconditioned solution will be discussed - the flow for the unpreconditioned case was visually similar.

The extent of the buffer zone relative to the full computational domain can be seen in Figure 4.74. In subfigure (a) the full computational domain is shown. An instantaneous solution in time is shown with the areas of the buffer zone shaded in grey. Subfigure (b) combines the full computational solution for density beneath the buffer zone contours. This shows how the vortical structures along the wall smoothly exit the wall without any reflections.
Figure 4.74. Effect of solution on buffer zone. (a) Computational solution excluding the buffer zone. (b) Solution in entire domain with buffer zone coefficients to show damping of buffer zone.

While the main comparisons are for the time averaged wall pressures on the cavity floor, the cavity flow solution will clearly affect the wall pressures. The nondimensional pressures and velocity magnitude (i.e. Mach number) at four different time steps are given below for both a preconditioned and unpreconditioned solution. See Figures 4.75 through 4.78. The solutions are qualitatively similar between the two solution methods with a vortex being generated at the cavity leading edge, traversing along the cavity opening, and then impinging on the downstream edge. The vortex then travels downstream along the wall. There is a difference between the preconditioned and unpreconditioned with regard to the location of the vortex as well as the magnitude of pressures in the domain. The preconditioned solution has a slightly higher pressure magnitude than the unpreconditioned solution. This is reflected in the time-averaged wall pressure fluctuations that are shown below.
**Figure 4.75.** $L/D = 6.25$ contours of nondimensional pressure at four different time steps using a preconditioned method. Each nondimensional time step is equal to $2.216\times10^{-4}$ seconds (15 nondimensional time steps = $3.324\times10^{-3}$ s).

**Figure 4.76.** $L/D = 6.25$ contours of nondimensional pressure at four different time steps using an unpreconditioned method. Each nondimensional time step is equal to $2.216\times10^{-4}$ seconds (15 nondimensional time steps = $3.324\times10^{-3}$ s).
Figure 4.77. $L/D = 6.25$ contours of Mach number at four different time steps using a preconditioned method. Each nondimensional time step is equal to 2.216E-4 seconds (15 nondimensional time steps = 3.324E-3 s).

Figure 4.78. $L/D = 6.25$ contours of Mach number at four different time steps using an unpreconditioned method. Each nondimensional time step is equal to 2.216E-4 seconds (15 nondimensional time steps = 3.324E-3 s).
4.2.4.2 Time-Averaged Wall Pressure Fluctuations

Stallings et al.'s wall pressure measurement locations for both cavity geometries are given in Figure 4.79. The measurement locations are spaced one inch apart lengthwise along the cavity floor. For the \( L/D = 5.42 \) cavity measurement locations #82 - #106 were recorded. The longer cavity, \( L/D = 6.25 \) includes measurement locations #107 - #110 as well. The pressure coefficient is defined as \( Cp = (p - p_{\text{inf}})/(0.5\rho U_{\text{inf}}) \).

![Figure 4.79. Locations for the wall pressure measurements.](image)

Figure 4.80 shows the time-averaged \( Cp \) on the cavity floor for the preconditioned and unpreconditioned solution as well as the measurements by Stallings et al. The character of the numerical solutions are similar with low pressure near the cavity leading edge, then an abrupt increase and then an almost constant pressure for the back half of the cavity. This is in contrast to the Stallings et al. measurement that has a very slightly decreasing pressure until the last 20% of the cavity is reached and then a significant increase. As was noted in the discussion on the flow results, the pressure magnitude for the preconditioned solution is greater than the unpreconditioned solution, which is reflected in the higher floor pressures.
Figure 4.80. Time-averaged wall pressure fluctuations measurements for $L/D = 6.25$ cavity.

To reconcile the discrepancy between the CHOPA solution and Stallings tal results other reasons for the difference were investigated. Stallings tal has a representation of the wall pressures on the cavity floor for different types of cavities. This is shown in Figure 4.81. Stallings describes the $L/D = 5.42$ and $L/D = 6.25$ as open cavities which is reflected in the measured cavity floor pressures. The cavity floor pressures predicted from CHOPA more closely resemble the transitional/closed flow cavity but that cavity floor pressure distribution is expected for cavities with $L/D = 13$. The reasons for the difference in cavity floor pressures is discussed further in Section 4.2.6.

4.2.5 L/D=5.42 Cavity Solution

This section discusses the CHOPA predictions for the $L/D = 5.42$ cavity for a Mach number of 0.2. Wall pressures were only simulated for a Mach number as low as 0.2 but the numerical results warranted an additional run to understand how the simulations were behaving. This will be discussed in more detail below. Unless otherwise specified all predictions shown were conducted with the preconditioned method. The timing statistics for the computations are also given to understand the effects of using a preconditioner.
Figure 4.81. Wall pressures on cavity floor for different length cavities. This is Figure 7 from Reference [3].

4.2.5.1 Flow Results

The wall pressure fluctuations on the cavity floor are recorded once the simulation has reached a quasi-steady state. As with the previous simulation the results from both a preconditioned and unpreconditioned solutions are presented.

The contours of nondimensional pressure and Mach number at four different time steps are shown in Figures 4.82 through 4.85. There is a similarity between the pressure for both solution methods. There is a vortex that is generated at the leading edge of the cavity that travels along the shear layer, impinges on the downstream cavity edge and is then ejected from the cavity. The vortex then travels along the wall before it smoothly exits the domain. The pressure inside the cavity is also slightly higher for the preconditioned method. The plots of nondimensional velocity magnitude, which is Mach number, also shows evidence of vortices being generated and convected downstream. This can be seen where there are regions of high velocity (e.g. red contours) next to low velocity (e.g. blue contours). This review of the cavity flow solutions did not reveal any anomalies with the solution.
4.2.5.2 Time-Averaged Wall Pressure Fluctuations

The wall pressure measurement locations are the same as the $L/D = 6.25$ cavity with the exception of points #107 - #110, as these are not available due to the small cavity length.

The time-averaged $C_p$ on the cavity floor for both solution methods and the measurements by Stallings et al. are given in Figure 4.86. The measurements by Stallings et al. look almost identical to those measured for the $L/D = 6.25$ cavity. The CHOPA numerical solutions also look similar to the predictions for the longer cavity. There is a significant disparity between the measurements and the predictions. The next section describes an investigation into reason(s) behind this disparity.

4.2.6 Wake Mode

An explanation for the difference between the wall pressure measurements by Stallings et al. and the CHOPA predictions required an investigation into the effects of two-dimensional versus three-dimensional cavity flows. Other researchers have examined the differences between two-dimensional and three-dimensional cavity flows, such as when the width of the cavity is such that end effects can be ne-
Figure 4.83. $L/D = 5.42$ contours of nondimensional pressure at four different time steps using an unpreconditioned method.

glected and the flow can be considered two-dimensional. Ahuja and Mendoza [225] showed that the cavity flow can be considered two-dimensional when the ratio of cavity width to cavity length, $W/L$, is greater than unity. There are notable three-dimensional effects when that ratio is below unity. Three-dimensional cavities, compared to two-dimensional cavities, typically have far field acoustic tone amplitudes as well as lower broadband noise.

Beyond characterization of cavity flows as acting as two- or three-dimensional cavities, the “wake mode” was discovered by Gharib and Roshko [4] when they were measuring the flow over an axisymmetric cavity. The cavity was created with a cutout in an ogive-shaped body so there are no end effects as there would be in a cavity cutout from a flat surface. They measured the cavity drag for a range of boundary layer momentum thicknesses, $\theta$, at the point of separation (i.e. cavity leading edge). When the ratio of cavity length to momentum thickness, $L/\theta$ was below 160 the cavity flow was in a low-drag regime and the flow behaved as expected. However, when the ratio was increased beyond 160, by reducing the incoming momentum thickness, the cavity drag increased significantly. Flow visualization of the shear layer indicated that for $L/\theta > 160$ the cavity flow became unstable on a scale greater than the diameter of the ogive model. The flow field more closely resembled the wake from a three-dimensional bluff body with the
cavity trailing edge influencing the wake within the shear layer. Hence, the cavity was considered to be in a wake mode.

With the appearance of a wake mode, the cavity drag increases significantly through the development of very large vortical structures inside the cavity. These vortical structures develop and span a considerable length of the cavity. These vortical structures are then quickly and violently ejected from the cavity. Plots of pressure coefficient along the cavity floor for two different $L/\theta$ ratios are reprinted from Gharib and Roshko’s report in Figure 4.87. Open circles are the pressures on the cavity downstream wall and solid symbols are the upstream wall. When the boundary layer thickness is reduced, and $L/\theta$ ratio increased as in the lower figure the pressure distribution on the aft wall increases significantly. This pressure distribution on the lower figure is the wake mode. The increase in the overall cavity drag seen in the wake mode is due to the large pressure difference between the pressures on the upstream and downstream wall.

Numerical studies on cavity flows were conducted well before Gharib and Roshko’s discovery, but afterwards researchers started to investigate differences between two- and three-dimensional cavity flows. As computational models of two-dimensional cavities can be used to more quickly gain an understanding of cavity flows and cavity flow tones, they have generated a significant body of work.
However, as the computational requirements for a three-dimensional cavity simulation have been reduced due to increases in computer processing power and speed, the comparisons between two-dimensional and three-dimensional cavities have been investigated by numerous other researchers.

The developers of the initial CHOPA code compared two- and three-dimensional cavity flows and acoustics [1]. They showed that for a shallow cavity with $L/D = 4.4$ the 2D cavity oscillated in a wake mode. The results showed that a large scale vortical structure developed on the cavity floor next to the upstream cavity wall. A pressure drop occurred on the upstream wall due to the flow separation of the vortex as it was ejected from the cavity. Thus the cavity flow impinged on the cavity floor and reattached there, instead of the downstream cavity edge. When the three-dimensional cavity flow was simulated there was no large scale vortical structure generated within the cavity and the pressure distribution was almost uniform between the upstream and downstream wall. Bres and Colonius [226] conducted a study to specifically investigate the cavity wake mode using Direct Numerical Simulation. This was a follow-on investigation into the wake mode in two-dimensional cavities [227]. In their work they showed that three-dimensionality of the cavity leads to the suppression of the wake mode. When random disturbances were added to the inflow velocity of a two-dimensional cavity that was oscillating in a wake mode.
mode, it caused the cavity to return to a shear layer mode of oscillation.

Another, more recent, investigation into the wake mode was undertaken by Zhang and Naguib [64]. A experimental study of an ogive-body, similar to the original work by Gharib and Roshko was conducted with much more extensive measurements. While their focus was on understanding sidewall effects on three-dimensional cavity flows, they were able to replicate the results of Gharib and Roshko and show that the wake mode exists for higher \( L/\theta \) ratios.

With the perspective from these researchers, the \( M = 0.2 \) results were reviewed again to see if a wake mode could be responsible for the differences between the Stallings tal measurements and the CHOPA solutions. For the \( L/D = 5.42 \) cavity the ratio of cavity length to momentum thickness is \( L/\theta = 433 \) while the same ratio is 500 for the \( L/D = 6.25 \) cavity. Both of these values are well beyond the minimum value of 160 that was given by Gharib and Roshko.

To ensure that the current version of CHOPA can accurately predict a cavity wake mode, the experiment of Plentovich [228] has been simulated. This is the same analysis that was conducted by Shieh and Morris [1] when they examined three-dimensionality effects. This is a \( L/D = 4.4 \) cavity at a Mach number of 0.6. Given the much higher Mach number than has been examined before in this study the unpreconditioned solution method is used as it has been shown

\[ \text{Figure 4.86. Time-averaged wall pressure fluctuations measurements for } L/D = 5.42 \text{ cavity for } M = 0.2. \]
preconditioning does not have a benefit for flow Mach numbers greater than $M = 0.4$. A comparison between the $\rho u$ streamlines from Shieh and Morris [1] and the current CHOPA version is shown in Figure 4.88. The results are almost identical indicating that the current version of CHOPA can still accurately predict cavity flows at higher Mach numbers.

To determine if the current Stallings simulation predicts a wake mode it would be best to compare the wall pressures on the upstream and downstream wall. However, since Stallings taldid not record pressures on the upstream or downstream cavity wall, simulation results on those walls were not obtained at those locations. Instead, time averaging of the cavity simulations was performed to determine the streamlines for the two cavity flow solutions. Time-averaged contours of $\rho u$ for the $L/D = 5.42$ solution are shown in Figure 4.89. The time averaging was done over one thousand time steps. The vortex within the cavity is quite large, with a length of almost half the cavity. This is a good indication that the cavity is oscillating in a wake mode. The same plots are given in Figure 4.90 for the $L/D = 6.25$ cavity.
Figure 4.88. Comparison of the time-averaged streamlines of $\rho u$ for Shieh and Morris (top left) and CHOPA (bottom right).

Again, the vortex inside the cavity is almost half the length of the cavity. The unpreconditioned solution vortex is slightly larger than the preconditioned solution but both results still indicate the strong possibility of a wake mode.

Figure 4.89. Time-averaged contours of $\rho u$ for $L/D = 5.42$ with Mach number = 0.2 flow.

While the $M = 0.2$ streamline results for both cavities point to a wake mode, an additional calculation has been performed to further validate this. The $L/D = 5.42$ cavity simulation is repeated for a lower Mach number, 0.1. This was not part Stallings et al.’s experiments, so no comparisons can be made, but this will provide a good indication if a wake mode is occurring. At a lower flow speed the $L/\theta$
ratio will be slightly lower. If there is a vortex that is about half the size of the cavity length then that will point to a wake mode. The time-averaged streamlines of nondimensional density times Mach number for the $M = 0.1$ solutions is shown in Figure 4.91. The time averaging is again over one thousand time steps. The preconditioned solution, which should perform well at this lower Mach number, has a vortex with a length of almost half the cavity. The unpreconditioned solution has two large vortices inside the cavity, one near the leading edge and one that envelopes the trailing edge. It is known at low Mach numbers that the artificial dissipation for a standard, i.e. unpreconditioned, solution method is often too high for an accurate solution. The non-physical addition of the artificial dissipation will dampen the shear layer and the flow will behave more like a driven cavity flow. This driven cavity flow results in large vortices in each corner of the cavity, which is shown here. Thus while the unpreconditioned solution does not accurately model the actual behavior the solution is behaving as expected.
4.2.7 Timing

The timing statistics for the computation will also be given to understand how the use of a preconditioner can affect the overall calculation time. The total calculation times for both cavity $L/D$ ratios are given in Figure 4.92. The total calculation time shown represents 1500 time steps for the $M = 0.2$ solution and 1000 time steps for the $M = 0.1$ solution. For the preconditioned solution method the portion of the total calculation time required to implement the preconditioner is shown in blue. From this graph it can be seen that the unpreconditioned solutions take less time than the preconditioned solution. The average number of subiterations for the $L/D = 5.42$ cavity is 3707 for preconditioned and 3781 unpreconditioned. For the $L/D = 6.25$ cavity the number of subiterations are 3655 and 3833 for preconditioned and unpreconditioned cases, respectively. Each subiteration calculation takes longer with the preconditioned method because of the additional calculations to implement the preconditioner. However, while the unpreconditioned solution takes less time, it is recommended to use a preconditioner while solving for low Mach number flows. Mach number flows below 0.4 will see a benefit in accuracy. Typically, as was shown for the $M = 0.1$ solution, a preconditioned method is required for flows below $M = 0.2$. For the $M = 0.1$ solution the unpreconditioned method takes less time but the solution is non-physical due to the extra artificial dissipation.

![Figure 4.92](image)

Figure 4.92. Total calculation times for the Stallings cavity simulations. The $M = 0.1$ solution is only for 1000 time steps.
4.2.8 Conclusions

In reviewing the Stallings tal cavity simulations, the data indicates to a wake mode that is responsible for the difference between the experimental and numerical results. These results show that care must be used when undertaking a two-dimensional simulation with regard to various parameters to ensure that the correct physical mechanism is predicted. The $W/L$ ratio for the Stallings cavities were also much less than unity indicating a highly three-dimensional flow, which can affect the wall pressures.
4.3 Conclusions

The preconditioner used with the CHOPA code is stable for cavity flow solutions as low as $M = 0.05$. It should be stable for much lower flow speeds but these were not tested in this work. When compared to the experiment of Block the numerical predictions for the cavity tone frequencies were more accurate for the deeper cavity, $L/D = 0.78$, than the shallow cavity, $L/D = 2.35$. For the deeper cavity the scaling with flow speed for the cavity tones, i.e. Strouhal scaling, was captured over the different simulations, but the transition to a Helmholtz scaling (i.e. acoustic response of the depth-mode of the cavity) was not captured. Using the preconditioned method for the deeper cavity, the total calculation time was inversely proportional to the Mach number except for the $M = 0.05$ calculation. That simulation required a greater number of subiterations for convergence and had a much higher calculation time. Without preconditioning a stable, physically-realistic solution is not expected for these flow speeds. The timing for the shallow cavity was relatively insensitive to flow speed, which is what would be expected for using a preconditioned method. The total calculation time for the $M = 0.05$ cavity was again greater than other flow speeds with a larger number of subiterations required for convergence.

The comparisons of the time-averaged wall pressures measured by Stallings tal to the preconditioned CHOPA simulation were not good but this is argued to be due to differences between two-dimensional (i.e. CHOPA) and three-dimensional (i.e. Stallings) cavity flows. In two-dimensional flows where the boundary layer is thin relative to the cavity length ($L/\theta > 160$) the cavity responds in a wake mode. A large vortex is generated that spans at least half of the cavity length and is then violently ejected from the cavity. This leads to low pressures on the upstream part of the cavity floor and higher pressures on the downstream part of the cavity floor. The development of the large vortex also leads to increased drag. This is significantly different than the three-dimensional flow and explains the difference between the cavity wall pressure comparisons. The total calculation times for the Stallings tal simulations were similar between Mach numbers but only $M = 0.1$ and $M = 0.2$ were simulated, so it is difficult to discern a trend. The unpreconditioned solution did have a lower calculation time but the solutions obtained at these lower Mach numbers were non-physical due to too much artificial dissipation.
Chapter 5  
Summary, Conclusions and Directions for Future Research

This chapter summarizes the contributions of this work in advancing the field of computational aeroacoustics. The tones and wall pressures generated by low speed hydrodynamic flow over a range of cavity geometries are simulated. The first section reviews the conclusions made based on the numerical simulations and then directions for future research are addressed.

5.1 Summary and Conclusions

The flow over cavities has historical importance in the aircraft industry with initial studies being undertaken to understand the physical mechanisms for the delivery of weapon stores from aircraft bomb bays. This type of work was for moderate to high subsonic Mach number flows but as the importance in reducing the noise generation from landing and taking off aircraft, low Mach number flows began to gain importance. In the low Mach number regime other applications have become important as well, from passenger comfort in automobiles with open windows or sun roofs to underwater vehicles. Underwater vehicles can be affected detrimentally by the generation of far field noise from hydrodynamically generated cavity flow as well as structural fatigue from unsteady wall pressures on the cavity. The focus of the present work is on the noise generated from low speed flows in-air, but all methodologies can be extended to almost incompressible flows such as water.

For hydrodynamic flow over a cavity there are two main mechanisms by which noise (both broadband and tonal) are generated and those mechanisms are often
segregated based on cavity geometry: shallow or deep. The dividing line between the types of cavities is generally considered $L/D = 1$. Shallow cavities have ratios greater than one and deep cavities have ratios less than one. A cavity tone can be generated for each type of cavity but the mechanisms are different, although both required a feedback loop and an oscillating shear layer.

For shallow cavities the shear layer passes over the upstream cavity edge where it separates. Instability waves that are inherent in the shear layer cause it to oscillate. The shear layer then impinges on the downstream cavity edge, which generates a pressure pulse that travels upstream to the separation edge and can impart energy to the separating shear layer. When the period of oscillation of the shear layer is an integer multiple of the sum of the transit times for the instability wave to traverse the cavity and the acoustic wave to propagate back, then a feedback loop is established. This is the most common mechanism for cavities at low to moderate subsonic Mach numbers. As the flow speed is increased a cavity tone can be generated just from impingement on the downstream cavity edge. These are Strouhal types of cavity response.

The forcing function for a deep cavity is again an oscillating shear layer but the frequency of oscillation matches the depth mode of the cavity. This is a Helmholtz type of response for the cavity where the mass in the cavity oscillates and produces a sharp tone. For low Mach number flows a cavity tone that radiates to the far field is more likely for a deep cavity than for a shallow cavity. This is because the radiation mechanism for a shallow cavity requires more energy in the shear layer, which is typically provided by the kinetic energy in the flow (i.e. flow speed).

With the importance of cavity tones to a range of industries the ability to determine when cavity tones can be generated has become important to engineers. Experimental measurements to investigate different configurations is often time consuming and expensive. For this reason numerical simulations have become more prevalent to determine if a design will generate unwarranted tones. Computational aeroacoustic (CAA) simulations combine a computational fluid dynamic simulation with the increased accuracy required to capture the small perturbations that are responsible for generating acoustic tones and broadband noise. A key component of CAA calculations requires a high-order spatial discretization to propagate the acoustic pressure as well as a time accurate methodology that enables an unsteady solution. There are also enhanced boundary conditions that are required to permit
both the fluid dynamic and acoustic pressures to smoothly exit the domain without spurious reflections.

The present work has combined all of the requirements for general CAA codes into the CHOPA code. It uses a formally fourth order accurate Dispersion Relation Preserving Scheme (DRP) for the spatial discretization. A dual-time stepping scheme is used where an explicit, fourth-order accurate Runge-Kutta method is used for the inner (e.g. steady-state) loop and a second-order accurate backward implicit method for the outer loop. This has been shown to be sufficient for the aeroacoustic problems considered in this work. Different boundary conditions were examined for the present work, but ultimately a combination of an extrapolation boundary condition with a buffer zone was selected to robustly allow fluid flow and acoustic waves out of the computational domain without spurious reflections.

While the field of CAA has matured over the past two decades there are added difficulties that require additional effort for the low Mach number flows of interest for this thesis. A significant difficulty with low Mach number CFD or CAA calculations is due to the large disparity between the acoustic and convective wave speeds. In a time accurate calculation the time step used to integrate the equations in time is proportional to the inverse of the acoustic wave speed. This results in a very small time step required for numerical stability. However, it also necessitates a large number of time steps for fluid particles moving at the convective wave speed. The end result is extremely long calculation times. Additionally, the solution for very low Mach numbers using standard scalar based artificial dissipation schemes for the spatial discretization can lead to numerical instability or overdamping of important high-wavenumber modes.

A time derivative preconditioner is used in the present work to equalize the acoustic and convective wave speeds. The preconditioner destroys the time accuracy of the governing differential equations, which is why a dual time step is used. The solution is broken up into a series of steady-state solutions that are tied together with the outer loop of the dual time step scheme (see Figure 2.1). Buelow’s extension of the Choi-Merkle preconditioner [103, 139] is used in this work. A matrix-based artificial dissipation is used to effectively scale the amount of artificial dissipation used in the spatial discretization to the flow speed, ensuring that important features of the flow are not damped.

A number of validation studies have been conducted with the CHOPA code
prior to the cavity simulations. First, the calculation of a pressure pulse was made to ensure the extrapolation boundary condition with the buffer zone would allow for acoustic pressure pulses to exit the domain without reflection. This was successful, without any reflections at the buffer zone or computational boundary.

Next, the preconditioner was validated. First an inviscid form of Buelow’s extension to the Choi-Merkle viscous preconditioner (CMv-B) was tested using a uniform flow problem. A random perturbation was added to a uniform velocity distribution inside the computational domain. The number of iterations required for convergence using a preconditioned method with scalar or matrix based artificial dissipation was compared to the unpreconditioned solution. As the Mach number was reduced the effectiveness of the preconditioner becomes greater. Eventually the unpreconditioned method is unable to conduct a stable analysis. This validation case showed that the preconditioner is effective for low Mach number flows, has increased accuracy, and reduces the calculation time. The preconditioner was then tested against the analytical solution for a Gaussian pulse propagation. The exact solution is for the linearized Euler equations so the inviscid form of the preconditioner was used. The comparison between the unpreconditioned, preconditioned, and exact solution was extremely good, again showing the validity of the preconditioned solver.

The last test of the preconditioner was for the viscous preconditioner for laminar flow over a flat plate. Mach numbers between 0.4 and 0.01 were simulated with good agreement between the preconditioned and exact solutions. The difference between the exact and preconditioned solution was more noticeable as the Mach number was reduced. However the small differences are not expected to affect the prediction of cavity tones.

Two-dimensional simulations of low to moderate Mach number cavity flows have been performed. Since different source mechanisms are responsible for deep and shallow cavities, both were examined. The comparisons were made for cavity tone frequencies as well as cavity wall pressures.

Two different cavity geometries were analyzed to compare against the experimental measurements of Block: $L/D = 0.78$ and $L/D = 2.35$. For each cavity a Mach number range of 0.05 to 0.4 was simulated. The cavity tone frequencies predicted with the preconditioned method for the $L/D = 0.78$ cavity had an acceptable comparison with the frequencies within 15% of the experiment except for
M = 0.4, where a solution was not obtained. As the flow speed was reduced the comparison improved with the best predictions at M = 0.15. The CHOP A code has difficulty in predicting the transition of the cavity response from a Strouhal type scaling (at higher Mach numbers) to a Helmholtz type scaling (at low Mach numbers) and to continued the prediction of a Strouhal type response, even at the lower Mach numbers. Examination of the predicted convective velocities showed that they were below the typical convective velocity of $k_v = 0.57$ but the lower convective velocity is not a reason for the discrepancy in cavity tone frequency. The other cavity geometry for which cavity tones were predicted was the $L/D = 2.35$ cavity. The frequency of the cavities tones predicted with CHOP A did not match the experimental measurements well, and in fact appeared to be almost independent of the Mach number. The flow solutions were indicative of expected cavity flow and did not give an indication of a potential problem. It is possible that the invariance of cavity tone frequency with flow speed is due to standing waves in the solution, though this was not visually evident.

The other two-dimensional cavity simulation performed was to compare the wall pressures of a low Mach number cavity flow with experimental measurements. The Stallings calculation was for two shallow cavities: $L/D = 5.42$ and $L/D = 6.25$ at a Mach number of 0.2. The results were not consistent with the experimental measurements but a review of results from other researchers showed that the reason was very likely due to two-dimensional effects. The two-dimensional cavity simulation had a $L/\theta$ ratio well beyond the 160 limit established by Gharib and Roshko for which the two-dimensional cavities will resonate in a wake mode. When the cavity wall pressures for the CHOP A analysis were compared to measurements by Gharib and Roshko and other numerical results the same trend for wall pressures were seen that indicated a wake mode. An additional, lower Mach number simulation was made for the $L/D = 5.24$ cavity at a Mach number of 0.1 to verify that the wake mode was present and its presence was confirmed. The wall pressures were justified based on the two-dimensional simulation. The preconditioner allowed for a more accurate solution at these lower Mach numbers than would be possible for an unpreconditioned solution.

In all of the simulations conducted for this work the preconditioner has been shown to be necessary for an accurate solution at low Mach numbers. Above a Mach number of 0.4 the preconditioner was not necessary and in fact resulted in
longer calculation times than if a standard solution was used. The matrix-based artificial dissipation of the preconditioner allowed for more accurate simulations by not damping important modes of the cavity flow.

5.2 Suggestions For Further Studies

The cavity simulations conducted in this work used a preconditioner to provide accurate solutions at low Mach numbers while also reducing the calculation time. However, the solutions using the preconditioner were unstable at Mach numbers greater than or equal to 0.4. It was shown for the solution of the $L/D = 0.78$ that lower levels of artificial dissipation were needed to obtain a better representation of the cavity flow. It is likely that the scaling parameters for the artificial dissipation ($d_6$) need to be increased for stable flow at the higher Mach numbers.

A review of the results for the Block experiments showed that the simulations had difficulty in predicting the shift in cavity tone mechanism from Strouhal scaling (at Mach numbers greater than about 0.2) to Helmholtz scaling (at Mach numbers less than 0.2). The Helmholtz scaling occurs when the shear layer instability is responsible for exciting a depth mode of the cavity and the mass of fluid within the cavity resonates as a single entity. While the deeper cavity in the Block analysis ($L/D = 0.78$) was analyzed with several different grid clusterings for the cavity it is possible that a finer grid spacing in the vertical direction is required to capture the depth mode response.

An obvious area to focus on for future research is extending the simulations to three dimensions. The preconditioner has been developed and given for three dimensions in this work. The extensions to boundary conditions, buffer zone, and preconditioner are straightforward but time consuming. With this extension studies examining two-dimensional versus three-dimensional effects, specifically the development of a wake mode, could be undertaken. Three-dimensional analyses would also increase the usefulness of the code to more realistic geometries.

The tones produced from hydrodynamic flow over and in a cavity can be significantly affected by the response of the internal cavity structures. The flow will cause the walls to change shape, altering the volume of the cavity and potentially changing the magnitude of the cavity tones while also shifting the frequencies. Additional resonances may be introduced due to the non-rigid nature of the cavity
walls that would not appear in the rigid wall cavity case. This is an important feature for the development of an engineering tool since the fluid-structure interaction is often significant to shift the cavity tone frequency away from that predicted with simple formulas matching the natural frequency of the cavity with shear layer instabilities. Instead of incorporating a complex and computationally expensive grid that actively adapts to moving boundaries this can be modeled to first order with an appropriate boundary condition. A multi-mass, multi-spring model that has seen successful implementation in other CFD codes has been developed by Flanagan et al. [229].

Finally, the cavity tones predicted in the present work are due to the near field acoustic pressures. However, of interest to both low flying aircraft and underwater vessels are the cavity tones in the far field. Some cavity tones may be evanescent and not propagate to the far field. The direct calculation of a far field cavity tone would require extending the grid at least fifty to one hundred times the cavity length in a hemispherical direction around the cavity. This would lead to a very large number of grid points in two dimensions but in three-dimensions this would be beyond most computing resources. Instead, a far field propagation method could be used that would solve for the fluid dynamic variables in a domain similar to what was considered in this work. There are several methods that exist, such as Ffowcs Williams-Hawkings (FW-H) or Kirchhoff-Helmholtz methods. The FW-H equation is an extension of Lighthill’s acoustic analogy that accounts for surfaces in arbitrary motion [230]. A more recent version of the FW-H equation has been used in the prediction of radiated noise using integrals that are formed on permeable surfaces that allow regions of high vorticity as well as acoustic disturbances to pass through the surfaces [231]. There are several permutations of the FW-H approach that improve speed or memory requirements. One such approach is that of Farassat and Succi [232] and Brentner and Farassat [233].
Appendix A
Governing Equations in Computational Space

For implementation in a CFD code the Navier-Stokes equations are transformed to a uniform grid in computational space. This greatly simplifies the solution since the spatial discretization stencils assume a rectangular, uniformly spaced grid [82]. The generic coordinate system is given in nondimensional form for both three and two dimensions in this Appendix.

A.1 Three Dimensions

The grid will be transformed to a generic coordinate system that uses the following variables,

\[
\begin{align*}
\xi &= \xi(x, y, z, t) \\
\eta &= \eta(x, y, z, t) \\
\zeta &= \zeta(x, y, z, t) \\
\tau &= \tau(t)
\end{align*}
\]  

(A.1) (A.2) (A.3) (A.4)

Although the transformation can be made for a time-dependent grid all that follows assumes the grid is stationary. The Jacobian of the transformation is then defined as,
\[ J = \det \frac{\partial (x,y,z)}{\partial (\xi, \eta, \eta)} = \begin{vmatrix} \xi_x & \xi_y & \xi_z \\ \eta_x & \eta_y & \eta_z \\ \zeta_x & \zeta_y & \zeta_z \end{vmatrix}^{-1} \]

(A.5)

where the subscript refers to a derivative with respect to that variable, i.e. \( x_\xi = \frac{\partial x}{\partial \xi} \).

Solving the determinant for each derivative results in the transformation metrics,

\[
\begin{align*}
\xi_x &= J (y_\eta z_\zeta - y_\zeta z_\eta), \\
\eta_x &= J (y_\zeta z_\eta - y_\eta z_\zeta), \\
\zeta_x &= J (y_\xi z_\eta - y_\eta z_\xi), \\
\xi_y &= J (x_\zeta z_\eta - x_\eta z_\zeta), \\
\eta_y &= J (x_\xi z_\zeta - x_\zeta z_\xi), \\
\zeta_y &= J (x_\eta z_\xi - x_\xi z_\eta), \\
\xi_z &= J (x_\eta y_\zeta - x_\zeta y_\eta), \\
\eta_z &= J (x_\xi y_\zeta - x_\zeta y_\xi), \\
\zeta_z &= J (x_\xi y_\eta - x_\eta y_\xi).
\end{align*}
\]

(A.6)

With the transformation Jacobian given as,

\[
J = \left[ x_\xi (y_\eta z_\zeta - y_\zeta z_\eta) - x_\eta (y_\zeta z_\eta - y_\eta z_\zeta) + x_\zeta (y_\xi z_\eta - y_\eta z_\xi) \right]^{-1}
\]

(A.7)

Once the above transformations are applied to the Navier-Stokes equations the equations in a generalized coordinate system can be given as,

\[
\frac{\partial Q}{\partial t} + \frac{\partial E_1}{\partial \xi} + \frac{\partial F_1}{\partial \eta} + \frac{\partial G_1}{\partial \zeta} = H
\]

(A.8)

\[
Q = \begin{cases} 
\rho \\
\rho u \\
\rho v \\
\rho w \\
\rho E_t
\end{cases}
\]

(A.9)

\[
E_1 = \frac{1}{J} \left[ (E - E_w) + (F - F_v) + (G - G_v) \right]
\]

(A.10)

\[
F_1 = \frac{1}{J} \left[ (E - E_w) + (F - F_v) + (G - G_v) \right]
\]

(A.11)

\[
G_1 = \frac{1}{J} \left[ (E - E_w) + (F - F_v) + (G - G_v) \right]
\]

(A.12)
\[
E = \begin{cases}
\rho U \\
\rho U u + \xi_x p' \\
\rho U v + \xi_y p' \\
\rho U w + \xi_z p' \\
(\rho E_t + p) U
\end{cases},
E_v = \begin{cases}
0 \\
\xi_x \tau_{xx} + \xi_y \tau_{xy} + \xi_z \tau_{xz} \\
\xi_x \tau_{xy} + \xi_y \tau_{yy} + \xi_z \tau_{yz} \\
\xi_x \tau_{xz} + \xi_y \tau_{yz} + \xi_z \tau_{zz} \\
[\xi_x (u \tau_{xx} + v \tau_{xy} + w \tau_{xz} + q_x) + \xi_y (u \tau_{xy} + v \tau_{yy} + w \tau_{yz} + q_y) + \xi_z (u \tau_{xz} + v \tau_{yz} + w \tau_{zz} + q_z)]
\end{cases}, (A.13)
\]

\[
F = \begin{cases}
\rho V \\
\rho V u + \eta_x p' \\
\rho V v + \eta_y p' \\
\rho V w + \eta_z p' \\
(\rho E_t + p) V
\end{cases},
F_v = \begin{cases}
0 \\
\eta_x \tau_{xx} + \eta_y \tau_{xy} + \eta_z \tau_{xz} \\
\eta_x \tau_{xy} + \eta_y \tau_{yy} + \eta_z \tau_{yz} \\
\eta_x \tau_{xz} + \eta_y \tau_{yz} + \eta_z \tau_{zz} \\
[\eta_x (u \tau_{xx} + v \tau_{xy} + w \tau_{xz} + q_x) + \eta_y (u \tau_{xy} + v \tau_{yy} + w \tau_{yz} + q_y) + \eta_z (u \tau_{xz} + v \tau_{yz} + w \tau_{zz} + q_z)]
\end{cases}, (A.14)
\]

\[
G = \begin{cases}
\rho W \\
\rho W u + \zeta_x p' \\
\rho W v + \zeta_y p' \\
\rho W w + \zeta_z p' \\
(\rho E_t + p) W
\end{cases},
G_v = \begin{cases}
0 \\
\zeta_x \tau_{xx} + \zeta_y \tau_{xy} + \zeta_z \tau_{xz} \\
\zeta_x \tau_{xy} + \zeta_y \tau_{yy} + \zeta_z \tau_{yz} \\
\zeta_x \tau_{xz} + \zeta_y \tau_{yz} + \zeta_z \tau_{zz} \\
[\zeta_x (u \tau_{xx} + v \tau_{xy} + w \tau_{xz} + q_x) + \zeta_y (u \tau_{xy} + v \tau_{yy} + w \tau_{yz} + q_y) + \zeta_z (u \tau_{xz} + v \tau_{yz} + w \tau_{zz} + q_z)]
\end{cases}, (A.15)
\]

Here the \( \prime \) refers to a perturbation quantity, see Section 2.4.4.5. Also, \( \rho, p, \) and \( E_t \) are the density, pressure, and total energy, respectively. The contravariant velocites are found in Equation (A.18).

\[
U = \xi_x u + \xi_y v + \xi_z w \quad (A.16)
\]

\[
V = \eta_x u + \eta_y v + \eta_z w \quad (A.17)
\]

\[
W = \zeta_x u + \zeta_y v + \zeta_z w \quad (A.18)
\]
The viscous terms are given by,

\[
\tau_{xx} = \frac{2}{3} \frac{M}{Re} \mu \left[ 2 \left( \xi_x \frac{\partial u}{\partial \xi} + \eta_x \frac{\partial u}{\partial \eta} + \zeta_x \frac{\partial u}{\partial \zeta} \right) - \left( \xi_y \frac{\partial v}{\partial \xi} + \eta_y \frac{\partial v}{\partial \eta} + \zeta_y \frac{\partial v}{\partial \zeta} \right) - \left( \xi_z \frac{\partial w}{\partial \xi} + \eta_z \frac{\partial w}{\partial \eta} + \zeta_z \frac{\partial w}{\partial \zeta} \right) \right] \tag{A.19}
\]

\[
\tau_{yy} = \frac{2}{3} \frac{M}{Re} \mu \left[ - \left( \xi_x \frac{\partial u}{\partial \xi} + \eta_x \frac{\partial u}{\partial \eta} + \zeta_x \frac{\partial u}{\partial \zeta} \right) + 2 \left( \xi_y \frac{\partial v}{\partial \xi} + \eta_y \frac{\partial v}{\partial \eta} + \zeta_y \frac{\partial v}{\partial \zeta} \right) - \left( \xi_z \frac{\partial w}{\partial \xi} + \eta_z \frac{\partial w}{\partial \eta} + \zeta_z \frac{\partial w}{\partial \zeta} \right) \right] \tag{A.20}
\]

\[
\tau_{zz} = \frac{2}{3} \frac{M}{Re} \mu \left[ - \left( \xi_x \frac{\partial u}{\partial \xi} + \eta_x \frac{\partial u}{\partial \eta} + \zeta_x \frac{\partial u}{\partial \zeta} \right) - \left( \xi_y \frac{\partial v}{\partial \xi} + \eta_y \frac{\partial v}{\partial \eta} + \zeta_y \frac{\partial v}{\partial \zeta} \right) + 2 \left( \xi_z \frac{\partial w}{\partial \xi} + \eta_z \frac{\partial w}{\partial \eta} + \zeta_z \frac{\partial w}{\partial \zeta} \right) \right] \tag{A.21}
\]

\[
\tau_{xy} = \tau_{yx} = \frac{M}{Re} \left( \xi_x \frac{\partial u}{\partial \xi} + \eta_x \frac{\partial u}{\partial \eta} + \zeta_x \frac{\partial u}{\partial \zeta} \right) \tag{A.22}
\]

\[
\tau_{yz} = \tau_{zy} = \frac{M}{Re} \left( \xi_y \frac{\partial v}{\partial \xi} + \eta_y \frac{\partial v}{\partial \eta} + \zeta_y \frac{\partial v}{\partial \zeta} \right) \tag{A.23}
\]

\[
\tau_{xz} = \tau_{zx} = \frac{M}{Re} \left( \xi_z \frac{\partial w}{\partial \xi} + \eta_z \frac{\partial w}{\partial \eta} + \zeta_z \frac{\partial w}{\partial \zeta} \right) \tag{A.24}
\]

The heat transfer terms are given by,

\[
q_x = \frac{1}{\gamma - 1} \frac{M \mu}{Re Pr} \left( \xi_x \frac{\partial T}{\partial \xi} + \eta_x \frac{\partial T}{\partial \eta} + \zeta_x \frac{\partial T}{\partial \zeta} \right) \tag{A.25}
\]

\[
q_y = \frac{1}{\gamma - 1} \frac{M \mu}{Re Pr} \left( \xi_y \frac{\partial T}{\partial \xi} + \eta_y \frac{\partial T}{\partial \eta} + \zeta_y \frac{\partial T}{\partial \zeta} \right) \tag{A.26}
\]

\[
q_z = \frac{1}{\gamma - 1} \frac{M \mu}{Re Pr} \left( \xi_z \frac{\partial T}{\partial \xi} + \eta_z \frac{\partial T}{\partial \eta} + \zeta_z \frac{\partial T}{\partial \zeta} \right) \tag{A.27}
\]
A.2 Two Dimensions

In two dimensions the equations are similar, with understandable simplifications. The transformation metrics are now,

\[ \xi_x = J y_\eta, \quad \eta_x = -J y_\xi \]
\[ \xi_y = -J x_\eta, \quad \eta_y = J x_\xi . \]  
(A.28)

with the transformation Jacobian shown in Equation (A.30).

\[ J = \left[ x_\xi y_\eta - x_\eta y_\xi \right]^{-1} \]  
(A.29)

Applying these transformations gives the Navier-Stokes equations in a generalized coordinate system as,

\[ \frac{\partial Q}{\partial t} + \frac{\partial E_1}{\partial \xi} + \frac{\partial F_1}{\partial \eta} = H. \]  
(A.30)

\[ Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho E_t \end{bmatrix}, \]  
(A.31)

\[ E_1 = \frac{1}{J} \left[ (E - E_v) + (F - F_v) \right] \]  
(A.32)

\[ F_1 = \frac{1}{J} \left[ (E - E_v) + (F - F_v) \right] \]  
(A.33)

\[ E = \begin{bmatrix} \rho U \\ \rho U u + \xi_x p' \\ \rho U v + \xi_y p' \\ (\rho E_t + p) U \end{bmatrix}, \quad E_v = \begin{bmatrix} 0 \\ \xi_x \tau_{xx} + \xi_y \tau_{xy} \\ \xi_x \tau_{xy} + \xi_y \tau_{yy} \\ [\xi_x (u \tau_{xx} + v \tau_{xy} + q_x) + \xi_y (u \tau_{xy} + v \tau_{yy} + q_y)] \end{bmatrix}. \]  
(A.34)
\[ F = \begin{cases} 
\rho V \\
\rho V u + \eta_x p' \\
\rho V v + \eta_y p' \\
(\rho E_t + p) V 
\end{cases}, \quad 
F_v = \begin{cases} 
0 \\
\eta_x \tau_{xx} + \eta_y \tau_{xy} \\
\eta_x \tau_{xy} + \eta_y \tau_{yy} \\
\left[ \eta_x \left( u \tau_{xx} + v \tau_{xy} + q_x \right) \right. \\
\left. + \eta_y \left( u \tau_{xy} + v \tau_{yy} + q_y \right) \right] 
\end{cases}, \quad (A.35) \]

The \(^t\) refers to a perturbation quantity. The contravariant velocities are found in Equation (A.38).

\[ U = \xi_x u + \xi_y v \quad (A.36) \]
\[ V = \eta_x u + \eta_y v \quad (A.37) \]

The viscous terms are given by,

\[ \tau_{xx} = \frac{2}{3} \frac{M}{Re^\mu} \left[ 2 \left( \xi_x \frac{\partial u}{\partial \xi} + \eta_x \frac{\partial u}{\partial \eta} \right) - \left( \xi_y \frac{\partial v}{\partial \eta} + \eta_y \frac{\partial v}{\partial \eta} \right) \right] \quad (A.38) \]
\[ \tau_{yy} = \frac{2}{3} \frac{M}{Re^\mu} \left[ 2 \left( \xi_y \frac{\partial v}{\partial \xi} + \eta_y \frac{\partial v}{\partial \eta} \right) - \left( \xi_x \frac{\partial u}{\partial \eta} + \eta_x \frac{\partial u}{\partial \eta} \right) \right] \quad (A.39) \]
\[ \tau_{xy} = \tau_{yx} = \mu \frac{M}{Re} \left[ \xi_y \frac{\partial u}{\partial \xi} + \eta_y \frac{\partial u}{\partial \eta} + \xi_x \frac{\partial v}{\partial \xi} + \eta_x \frac{\partial v}{\partial \eta} \right] \quad (A.40) \]

The heat transfer terms are given by,

\[ q_x = \frac{1}{\gamma - 1} \frac{M}{Re \, Pr} \left( \xi_x \frac{\partial T}{\partial \xi} + \eta_x \frac{\partial T}{\partial \eta} \right) \quad (A.41) \]
\[ q_x = \frac{1}{\gamma - 1} \frac{M}{Re \, Pr} \left( \xi_y \frac{\partial T}{\partial \xi} + \eta_y \frac{\partial T}{\partial \eta} \right). \quad (A.42) \]
Appendix B
Dispersion Relation Preserving Coefficients

The coefficients for a seven point stencil based on the DRP scheme are given below. This scheme assumes the point of interest is at $i = 0$. They are derived from Lockard et al. [234].

\[
\begin{align*}
    a_{-3} &= -0.0193245151212886147142 \\ 
    a_{-2} &= 0.1606313938184877921905 \\ 
    a_{-1} &= -0.7632892422731097402381 \\ 
    a_0 &= 0.0 \\ 
    a_1 &= 0.7632892422731097402381 \\ 
    a_2 &= -0.1606313938184877921905 \\ 
    a_3 &= 0.0193245151212886147142
\end{align*}
\]
## Appendix C

### Preconditioner Information

Table C.1. Definition of various parameters associated with the preconditioners.

<table>
<thead>
<tr>
<th>Preconditioner</th>
<th>Subscript</th>
<th>Equation Set</th>
<th>Solution Vector</th>
<th>References</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turkel</td>
<td>TI</td>
<td>Incompressible</td>
<td>$Q_1$</td>
<td>[106]</td>
</tr>
<tr>
<td>Turkel87</td>
<td>T87</td>
<td>Euler</td>
<td>$dQ_{ii}$</td>
<td>[106, 110, 147]</td>
</tr>
<tr>
<td>Lee87</td>
<td>Lee87</td>
<td>Euler</td>
<td>$dQ_{ii}$</td>
<td>[147]</td>
</tr>
<tr>
<td>Modified Turkel</td>
<td>MT</td>
<td>Euler</td>
<td>$dQ_{ii}$</td>
<td>[152]</td>
</tr>
<tr>
<td>Turkel Comp./Incomp.</td>
<td>TC/1</td>
<td>Euler</td>
<td>$Q'$</td>
<td>[114]</td>
</tr>
<tr>
<td>Turkel93</td>
<td>T93</td>
<td>Euler</td>
<td>$Q_S$</td>
<td>[107, 153]</td>
</tr>
<tr>
<td>Chorin/Turkel</td>
<td>C/T</td>
<td>Euler</td>
<td>$dQ_{ii}$</td>
<td>[152]</td>
</tr>
<tr>
<td>van Leer Optimal</td>
<td>opt</td>
<td>Euler</td>
<td>$dQ_{ii}$</td>
<td>[115]</td>
</tr>
<tr>
<td>van Leer stream-aligned</td>
<td>VLR91</td>
<td>Euler</td>
<td>$dQ_{ii}$</td>
<td>[129]</td>
</tr>
<tr>
<td>van Leer cartesian</td>
<td>VLR91</td>
<td>Euler</td>
<td>$dQ$</td>
<td>[154]</td>
</tr>
<tr>
<td>van Leer stream-aligned sub-optimal</td>
<td>VLR91sub</td>
<td>Euler</td>
<td>$dQ_{ii}$</td>
<td>[147]</td>
</tr>
<tr>
<td>van Leer 96</td>
<td>VL96</td>
<td>Euler</td>
<td>$dQ_{ii}$</td>
<td>[138, 155]</td>
</tr>
<tr>
<td>Choi Merkle inviscid</td>
<td>CMi</td>
<td>Euler</td>
<td>$Q$</td>
<td>[156, 157]</td>
</tr>
<tr>
<td>D. Lee stream-aligned</td>
<td>DL$[[i]]$</td>
<td>Euler</td>
<td>$dQ_{ii}$</td>
<td>[110, 147]</td>
</tr>
<tr>
<td>D. Lee cartesian</td>
<td>DL</td>
<td>Euler</td>
<td>$dQ$</td>
<td>[147]</td>
</tr>
<tr>
<td>Jacobi</td>
<td>J</td>
<td>Euler</td>
<td>$dQ$</td>
<td>[161]</td>
</tr>
<tr>
<td>Turkel viscous Jacobi</td>
<td>TVJ</td>
<td>Navier-Stokes</td>
<td>$Q_S$</td>
<td>[136]</td>
</tr>
<tr>
<td>VLR viscous Jacobi</td>
<td>V-VJ97</td>
<td>Navier-Stokes</td>
<td>$dQ_{ii}$</td>
<td>[138]</td>
</tr>
<tr>
<td>VLR viscous Jacobi</td>
<td>V-VJ99</td>
<td>Navier-Stokes</td>
<td>$dQ_{ii}$</td>
<td>[180]</td>
</tr>
<tr>
<td>Pierce/Giles</td>
<td>PG</td>
<td>Navier-Stokes</td>
<td>Arbitrary</td>
<td>[162]</td>
</tr>
<tr>
<td>Lee dispersion analysis</td>
<td>Lee-NS</td>
<td>Navier-Stokes</td>
<td>$dQ_{ns_{ii}}$</td>
<td>[147]</td>
</tr>
</tbody>
</table>
The following definitions are used to define the solution vector:

\[
\begin{align*}
\mathbf{dQ}_{||} &= (dp/\rho c, dq, dr, ds, dp - c^2 dp)^T \\
\mathbf{dQ}_{\text{ns}} &= (a dp/(\sqrt{\gamma} \rho), dq, dr, ds, a dT/(\sqrt{\gamma} (\gamma - 1) T))^T \\
\mathbf{dQ} &= (dp/\rho c, du, dv, dS)^T \\
\mathbf{Q} &= (\rho, \rho u, \rho v, \rho w, E_t)^T \\
\mathbf{Q}_1 &= (p, u, v)^T \\
\mathbf{Q}_2 &= (p_1/\beta, u, v, T)^T \\
\mathbf{Q}_S &= (p, u, v, w, S)^T \\
\mathbf{Q}_v &= (p, u, v, w, T)^T \\
\mathbf{Q}' &= (p', \rho u, \rho v, E')^T
\end{align*}
\]
Appendix D
Preconditioner Matrices

Preconditioners are in the form of,

\[ \Gamma \frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = 0. \quad (D.1) \]

Note that preconditioners designed to act upon the residual instead of the time derivative and are noted by \( \Gamma^{-1} \). It does not matter if the time derivative or the residual is preconditioned as these two methods are equivalent since,

\[ \Gamma \frac{\partial A}{\partial t} + \frac{\partial B}{\partial x} = 0 \iff \frac{\partial A}{\partial t} + \Gamma^{-1} \frac{\partial B}{\partial x} = 0 \quad (D.2) \]

As these two methods are interchangeable care must be taken when reviewing a preconditioner matrix to be sure which part of the system the preconditioner is designed to affect. Preconditioners will often be presented in symmetrizing variables for a stream-aligned coordinate system for ease of viewing and analysis. If additional solution variables are required please consult the transformation methods in [153].

D.1 Incompressible Preconditioners

Turkel

Name: TI [106, 142]
Solution vector: \( Q_v = (p, u, v)^T \) (Energy equation is decoupled from others)
\[ \Gamma_{TI}^{-1} = \begin{bmatrix} \frac{1}{\beta^2} & 0 & 0 \\ \frac{\alpha u}{\beta^2} & 1 & 0 \\ \frac{\alpha v}{\beta^2} & 0 & 1 \end{bmatrix} \]  

(D.3)

Preconditioning parameters: \( \alpha, \beta \)

\[ \beta = \begin{cases} \max[(2 - \alpha)q^2, \varepsilon] & \alpha < 1 \\ K \max[\alpha q^2, \varepsilon] & \alpha \geq 1 \end{cases} \]  

(D.4)

Where \( K \) is a value slightly greater than one and \( \varepsilon \) is a small percentage of the maximum flow velocity squared, \( q^2 \) and \( \alpha = 1 \) give the optimal condition number [106]. This has been extended by Fiedler and Ashcroft [142]. Djeddi et al. took the preconditioning matrix from Turkel and rederived the matrix to include the Spalart-Allmaras turbulence model.

\section*{D.2 Euler Preconditioners}

\subsection*{D.2.1 Turkel Preconditioners}

Turkel87 Symmetry variables

Name: T87 [106, 110, 147]

Solution vector: \( dQ_{||} = (dp/\rho c, dq, dr, ds, dp - c^2 d\rho)^T \)

\[ \Gamma_{T87}^{-1} = \begin{bmatrix} \beta^2 & 0 & 0 & 0 & 0 \\ -\alpha M & 1 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 1 \end{bmatrix} \]  

(D.5)

Preconditioning parameters:
\[ \alpha = 1 + M^2 \]
\[ \beta^2 = M^2 \]  \hspace{1cm} (D.6)
\[ \beta^2 > \alpha M^2 \] \hspace{1cm} (Required for symmetry)

Indication is given by Lee [147] that this preconditioner may result in numerical difficulties as written. They offer a similar matrix that removes the difficulty with \( \beta = M \).

\[
\Gamma_{Lee87}^{-1} = \begin{bmatrix}
\frac{M^2}{\beta} & 0 & 0 & 0 \\
0 & \frac{M}{\beta} & 1 & 0 \\
0 & 0 & \beta & 0 \\
0 & 0 & 0 & 1
\end{bmatrix}
\]  \hspace{1cm} (D.7)

**Modified Turkel**

Name: MT87 [152]
Solution vector: \( dQ_{||} = (dp/\rho c, dq, dr, dp - c^2 d\rho)^T \)

\[
\Gamma_{MT87}^{-1} = \begin{bmatrix}
\beta^2 & 0 & 0 & 0 \\
-(1 + \beta^2)M & 1 & 0 & 0 \\
0 & 0 & 1 & 0 \\
0 & 0 & 0 & 1
\end{bmatrix}
\]  \hspace{1cm} (D.8)

Preconditioning parameters:

\[ \beta^2 = \frac{M^2}{1 - M^2} + \eta M_{\infty}^2 \]  \hspace{1cm} (D.9)

Where \( \eta \) is problem dependent but \( O(10^{-1}) \). This is the same as Turkel87 with \( \alpha = 1 + \beta^2 \). For symmetrizing variables it is required that \( \beta^2 > \alpha M^2 \) so this is used as a constraint to develop the values of \( \beta \),

\[ \beta^2 > \frac{M^2}{1 - M^2} \]  \hspace{1cm} (D.10)
Turkel C/I

Name: TC/I [114]
Solution vector: \( Q = (p', \rho u, \rho v, E')^T \)

\[
\Gamma^{-1}_{TC/I} = -[I] - \frac{\Delta}{R + h_\infty} \begin{bmatrix}
(R + h_\infty)(1 - \frac{1}{\Delta}) & -u & -v & 1 \\
(R + h_\infty)(-uB_2) & u^2B_2 & uvB_2 & -uB_2 \\
(R + h_\infty)(-vB_2) & uvB_2 & v^2B_2 & -vB_2 \\
(R + h_\infty)(-B_3) & uB_3 & vB_3 & -B_3
\end{bmatrix}
\] (D.11)

where

\[
p' = p - p_\infty \\
E' = c_p \rho (T - T_\infty) - p' + \rho q/2 \\
\Delta = (R + h_\infty)\beta^2 / h \\
R = q^2 / 2 \\
h = c_p T = c^2 / (\gamma - 1)
\]

Preconditioning parameters [114]:

\[
\beta^2 = \min \left\{ \max \left[ \beta_1 (u^2 + v^2), \beta_{\min}^2 \right], c^2 \right\} \\
\alpha = \min \left[ 1, \frac{\beta^2}{\beta_1 (u^2 + v^2)} \right]
\]

\( \beta_{\min} \) is varied from 0 to 0.4 and \( \beta_1 \) is \( \mathcal{O}(10^{-1}) \), based on numerical experiments.

Turkel 93

Name: T93 [107, 153]
Solution vector: \( Q = (p, u, v, w, S)^T \).
\[ \Gamma_{T93} = \begin{bmatrix} \frac{c^2}{\beta^2} & 0 & 0 & 0 & \delta \\ \frac{\alpha u}{\rho \beta^2} & 1 & 0 & 0 & 0 \\ \frac{\alpha v}{\rho \beta^2} & 0 & 1 & 0 & 0 \\ \frac{\alpha w}{\rho \beta^2} & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 & 0 \end{bmatrix} \] (D.12)

Preconditioning parameters [114, 153]:

\[
\begin{align*}
\delta &= 0 \\
\alpha &= \min \left[ 1, \frac{\beta^2}{\beta_1(u^2 + v^2 + w^2)} \right] \\
\beta^2 &= \min \left\{ \max \left[ K_1 q^2 \left( 1 + \frac{1 - M_0^2}{M_0^2} M^2 \right), K_2 q_\infty^2 \right], c^2 \right\}
\end{align*}
\] (D.13) (D.14) (D.15) (D.16)

where \( M_0 \) is a cutoff Mach number above which it is not preconditioned. Typically \( K_1 \) is between 1 and 1.1, \( K_2 \) is between 0.4 and 1, and \( \beta_1 \) is \( O(10^{-1}) \).

**Chorin/Turkel**

Name: C/T [152]

Solution vector: \( dQ_{||} = (\frac{du}{pc}, dq, dr, ds, dp - c^2 d\rho)^T \)

\[ \Gamma_{CT}^{-1} = \begin{bmatrix} \beta^2 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 1 \end{bmatrix} \] (D.17)

Preconditioning parameters:

\[
\begin{align*}
\beta &= M 
\end{align*}
\] (D.18) (D.19)
### D.2.2 van Leer Preconditioners

**Optimal**

Name: Optimal [115]
Solution vector: \( \mathbf{dQ}_\parallel = (\frac{dp}{\rho c}, dq, dr, dp - c^2 d\rho)^T \)

\[
\Gamma_{opt} = \begin{bmatrix}
\frac{1}{M^2} - \frac{\beta^2 s \cos \theta}{M} & \frac{1}{M} & - \beta s \sin \theta & 0 \\
\frac{1}{M} & 1 & 0 & 0 \\
\beta s \sin \theta & 0 & -M s \cos \theta & 0 \\
0 & 0 & 0 & 1
\end{bmatrix}
\]  

(D.20)

where \( \theta \) and \( s \) are arbitrary.

**VLR91 Stream-aligned**

Name: VLR91 [129]
Solution vector: \( \mathbf{dQ}_\parallel = (\frac{dp}{\rho c}, dq, ds, dr, dp - c^2 d\rho)^T \)

\[
\Gamma^{-1}_V_{LR91} = \begin{bmatrix}
\frac{\tau}{\beta^2} M^2 & -\frac{\tau}{\beta^2} M & 0 & 0 & 0 \\
-\frac{\tau}{\beta^2} M & \frac{\tau}{\beta^2} + \varepsilon & 0 & 0 & 0 \\
0 & 0 & \tau & 0 & 0 \\
0 & 0 & 0 & \tau & 0 \\
0 & 0 & 0 & 0 & \varepsilon
\end{bmatrix}
\]  

(D.21)

Preconditioning parameters [154, 165, 235]:

\[
\beta = \sqrt{|1 - M^2|} \\
\tau = \min(\beta, \beta/M) \\
\varepsilon = \tau \left(1 + \frac{AR}{\beta}\right)
\]  

(D.22)

where \( \varepsilon \) is to handle rectangular cells. For a square cell \( \varepsilon = 1 \) gives the optimal spread of the characteristic wave speeds.
VLR91 Cartesian

Name: VLR91 [154]
Solution vector: \( dQ = \left( \frac{dp}{pc}, du, dv, dS \right)^T \)

\[
\Gamma_{VLR91}^{-1} = \begin{bmatrix}
\frac{M^2}{\beta} & -\frac{M}{\beta} \cos \theta & -\frac{M}{\beta} \sin \theta & 0 \\
-\frac{M}{\beta} \cos \theta \left( \frac{1}{\beta} + 1 \right) \cos^2 \theta + \beta \sin^2 \theta & \frac{1}{\beta} + 1 - \beta & \frac{1}{\beta} \sin \theta \cos \theta & 0 \\
-\frac{M}{\beta} \sin \theta \left( \frac{1}{\beta} + 1 - \beta \right) \sin \theta \cos \theta & \frac{1}{\beta} + 1 & \beta \cos \theta & 0 \\
0 & 0 & 0 & 1 
\end{bmatrix}
\]

Preconditioning parameters:

\[
\beta = \sqrt{|1 - M^2|} \tag{D.23}
\]

VLR91 Stream-aligned Sub-optimal

Name: VLR91sub [147]
Solution vector: \( dQ_{\parallel} = \left( \frac{dp}{pc}, dq, dr, dp - c^2d\rho \right)^T \)

\[
\Gamma_{VLR91sub}^{-1} = \begin{bmatrix}
\alpha \frac{M^2}{\beta} & -\alpha \frac{M}{\beta} & 0 & 0 \\
-\alpha \frac{M}{\beta} & \alpha \left( \frac{1}{\beta} + 1 \right) & 0 & 0 \\
0 & 0 & \beta & 0 \\
0 & 0 & 0 & \alpha 
\end{bmatrix} \tag{D.24}
\]

Preconditioning parameters [147]:

\[
\alpha = \frac{1}{2}
\]

\[
\beta = \sqrt{|1 - M^2|} \tag{D.25}
\]

VL96

Name: VL96 [138,155]
Solution vector: \( dQ_{\parallel} = \left( \frac{dp}{pc}, dq, dr, ds, dp - c^2d\rho \right)^T \)
Preconditioning parameters: \( \beta \)
\[ \Gamma_{VL96}^{-1} = \begin{bmatrix} \frac{M^2}{\beta} & -\frac{M}{\beta} f & 0 & 0 & 0 \\ -\frac{M}{\beta} & 1 + \frac{L}{\beta} & 0 & 0 & 0 \\ 0 & 0 & \beta & 0 & 0 \\ 0 & 0 & 0 & \beta & 0 \\ 0 & 0 & 0 & 0 & 1 \end{bmatrix} \] (D.26)

where

\[ f = \frac{\beta - 2M^2}{1 + M^2} \] (D.27)

Preconditioning parameters:

\[ \beta = \sqrt{|1 - M^2|} \] (D.28)

**D.2.3 Choi Merkle Inviscid**

Name: CMi [156, 157]
Solution vector: \( \mathbf{Q} = (\rho, \rho u, \rho v, \rho w, E_t)^T \)
Preconditioning parameters: N/A

\[ \Gamma_{CMi} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 \\ \frac{q^2}{2} \left( \frac{1}{M^2} - 1 \right) u \left( 1 - \frac{1}{M^2} \right) v \left( 1 - \frac{1}{M^2} \right) w \left( 1 - \frac{1}{M^2} \right) \end{bmatrix} \] (D.29)

**D.2.4 D. Lee Preconditioner**

Cartesian
Name: DL [110, 147]
Solution vector: \( d\mathbf{Q} = (\frac{d\rho}{\rho c}, du, dv, dS)^T \)
\[
\Gamma_{DL}^{-1} = \begin{bmatrix}
A_0 & \pm D \cos \psi & \pm D \sin \psi & 0 \\
\mp D \cos \psi & \varepsilon \sin^2 \psi + E \cos^2 \psi & -C \cos \psi \sin \psi & 0 \\
\mp D \sin \psi & -C \cos \psi \sin \psi & \varepsilon \cos^2 \psi + E \sin^2 \psi & 0 \\
0 & 0 & 0 & 1
\end{bmatrix}
\]  
(D.30)

where \( C = \varepsilon + A_0 - B_0, \) \( D = \sqrt{A_0 C}, \) \( E = B_0 - A_0. \)

Preconditioning parameters:

\[
A_0 = \begin{cases} 
\frac{M^2}{\varepsilon} & M < 1 \\
1 & M \geq 1
\end{cases} \quad \text{(D.31)}
\]

\[
B_0 = \begin{cases} 
0 & M < 1 \\
2\sqrt{\frac{1}{M^2} - 1} & M \geq 1
\end{cases} \quad \text{(D.32)}
\]

\[
\varepsilon = \tau \left( 1 + \frac{AR}{\beta} \right) \\
\tau = \beta \quad \text{(for } M < 1) \quad \text{(D.33)}
\]

\[
\beta = \sqrt{|1 - M^2|}
\]

**Stream-aligned**

Name: DL [110, 147]
Solution vector: \( dQ_{||} = (\frac{dp}{\rho c}, dq, dr, dS)^T \)

\[
\Gamma_{DL||}^{-1} = \begin{bmatrix}
\beta^2 & \pm \beta \sqrt{1 + \beta^2} & 0 & 0 \\
\mp \sqrt{1 + \beta^2} & -\beta^2 & 0 & 0 \\
0 & 0 & 1 & 0 \\
0 & 0 & 0 & 1
\end{bmatrix}
\]  
(D.34)

Preconditioning parameters:

\[
\beta = M \quad \text{(D.35)}
\]
D.2.5 Jacobian Preconditioner

Name: Jacobian [122, 160, 161]
Solution vector: \( dQ = (\frac{dp}{\rho c}, du, dv, dS)^T \)

\[
\Gamma^{-1} = c(|A| + |B|)^{-1} = \frac{1}{M} \begin{bmatrix}
\frac{M}{1+\beta^2} & -\frac{M^2}{1+\beta^2} & 0 & 0 \\
-M^2 & \frac{2M}{1+\beta^2} & 0 & 0 \\
0 & 0 & \frac{M}{M+1} & 0 \\
0 & 0 & 0 & 1
\end{bmatrix}
\] (D.36)

D.3 Navier-Stokes Preconditioners

The preconditioned Navier-Stokes can also be written in nonconservative form as,

\[
\Gamma \frac{\partial Q}{\partial t} + A_i \frac{\partial Q}{\partial x} + B_i \frac{\partial Q}{\partial y} + C_i \frac{\partial Q}{\partial z} - D_v \frac{\partial Q}{\partial x} - E_v \frac{\partial Q}{\partial y} - F_v \frac{\partial Q}{\partial z} = 0
\] (D.37)

where \( A_i, B_i, C_i \) are flux Jacobians and \( D_v, E_v, F_v \) are viscous Jacobians [22].

D.3.1 Viscous Jacobi Preconditioner

D.3.1.1 Turkel Preconditioner

Name: TVJ [136]
Solution vector: \( Q = (p, u, v, w, S)^T \)

\[
\Gamma_{TVJ} = \Gamma_{T93} + 2 \left( \frac{D_v}{\Delta x^2} + \frac{E_v}{\Delta y^2} + \frac{F_v}{\Delta z^2} \right)
\] (D.38)

D.3.1.2 van Leer Preconditioner

VLR Viscous Jacobian Preconditioner

Name: V-VJ97 [138]
Solution vector: \( dQ_{||} = (\frac{dp}{\rho c}, dq, dr, dp - c^2 d\rho)^T \)

227
\[
\Gamma_{V-V_j97} = \Gamma_{Eu} + \frac{1}{\alpha} \left( \frac{2D_v}{\Delta x^2} + \frac{2E_v}{\Delta y^2} - \Delta t \frac{\partial H}{\partial Q} \right)
\]  \hspace{1cm} \text{(D.39)}

where \( \Gamma_{Eu} \) is the VLR91 preconditioner.

Preconditioning parameters: \( \alpha, \beta \)

\[
\alpha = \frac{q(\beta + A\mathcal{R})}{\Delta x} \\
\beta = \sqrt{|1 - M^2|}
\]  \hspace{1cm} \text{(D.40)}

**VLR Viscous Jacobian Preconditioner**

Name: V-VJ99 [180]
Solution vector: \( dQ|| = (\frac{dp}{\rho c}, dq, dr, dp - c^2 dp) \)

\[
\Gamma_{V-V_j99} = \Gamma_{Eu-mod} + \frac{2}{\Delta x (1 - \kappa)} \left( D + A\mathcal{R}F \right)
\]  \hspace{1cm} \text{(D.41)}

where \( \kappa \) is a MUSCL (Monotone Upstream-centered Schemes for Conservation Laws) coefficient between -1 (full upwind) and 1 (central difference) that controls upwind biasing, \( D \) and \( E \) are the viscous Jacobians, and \( \Gamma_{Eu} \) is a modified VLR91 preconditioner based on Lee [147],

\[
\Gamma_{Eu-mod}^{-1} = \begin{bmatrix}
\xi \frac{M^2}{\beta^2} & -\xi \frac{M}{\beta^2} & 0 & 0 \\
-\xi \frac{M}{\beta^2} & 1 + \frac{\xi}{\beta^2} & 0 & 0 \\
0 & 0 & \xi & 0 \\
0 & 0 & 0 & 1
\end{bmatrix}
\]  \hspace{1cm} \text{(D.42)}

with \( \xi = \beta/(\beta + A\mathcal{R}_q) \) and the aspect ratio is defined as,

\[
A\mathcal{R}_q = \frac{\sum_i |u\Delta x_i + v\Delta y_i|}{\sum_i |v\Delta x_i - u\Delta y_i|}
\]  \hspace{1cm} \text{(D.43)}

where \( i \) is a cell face.

Preconditioning parameters:
\beta = \sqrt{|1 - M^2|} \quad (D.44)

**D.3.1.3 Pierce/Giles Preconditioner**

Name: PG [162]
Solution vector: Arbitrary
Preconditioning parameters: N/A

\[
\Gamma_{PG} = \frac{1}{\text{CFL}_H} \left( \frac{|A_i|}{\Delta x} + \frac{|B_i|}{\Delta y} + \frac{|C_i|}{\Delta z} + \frac{2D_v}{\Delta x^2} + \frac{2E_v}{\Delta y^2} + \frac{2F_v}{\Delta z^2} \right) \quad (D.45)
\]

where \text{CFL}_H is the hyperbolic CFL number corresponding to the stability region along the imaginary axis.

**D.3.2 Dispersion Analysis Preconditioner**

**D.3.2.1 Lee Preconditioner**

Name: Lee-NS [147]
Solution vector: \(dQ_{||} = (a \, dp/\sqrt{\gamma \rho}, dq, dr, ds, a \, dT/\sqrt{\gamma(\gamma - 1)T})^T\)

\[
\Gamma_{Lee-NS} = \Gamma_{Eu} + \frac{2}{\alpha} \left( \frac{D_v}{\Delta x^2} + \frac{E_v}{\Delta y^2} \right) \quad (D.46)
\]

with the modified VLR91 preconditioner becoming,

\[
\Gamma_{Eu} = \begin{bmatrix}
\frac{\gamma}{\beta \sigma} \sigma & -\frac{\gamma}{\beta \sigma} M & 0 & 0 \\
-\frac{\gamma}{\beta \sigma} M & \frac{\gamma}{\beta \sigma} + 1 & 0 & 0 \\
0 & 0 & \tau & 0 \\
0 & 0 & 0 & 1
\end{bmatrix} \quad (D.47)
\]

Preconditioning parameters:

\[
\alpha = \frac{\sqrt{2 \sigma}}{\Delta x} \quad (D.48)
\]
\[ \sigma = \begin{cases} \frac{M^2}{\text{Re}}, & \text{Re} \gg 1, \frac{M^2}{\text{Re}} \ll 1 \\ \frac{1}{\text{Re}} , & \text{Re} \ll 1, \frac{M^2}{\text{Re}} \gg 1 \end{cases} \] (D.49)

**D.3.2.2 Venkateswaran Merkle Preconditioner**

Name: VM [166]
Solution vector: \( Q_v = (p, u, v, w, T) \)

\[
\Gamma_{VM} = \begin{bmatrix}
\frac{1}{\varepsilon c^2} & 0 & 0 & 0 & 0 \\
\frac{u}{\varepsilon c^2} & \rho & 0 & 0 & 0 \\
\frac{v}{\varepsilon c^2} & 0 & \rho & 0 & 0 \\
\frac{w}{\varepsilon c^2} & 0 & 0 & \rho & 0 \\
\frac{h + \frac{1}{2}q^2}{\varepsilon c^2} - \delta & \rho u & \rho v & \rho w & \rho C_p
\end{bmatrix} \] (D.50)

\[ \varepsilon = \begin{cases} M^2, & \text{Re} \Delta x \gg 1 \\ \frac{M^2}{\Delta x}, & \text{Re} \ll 1, \frac{M^2}{\Delta x} \ll 1 \\ \frac{1}{\Delta x}, & \text{Re} \ll 1, \frac{M^2}{\Delta x} \gg 1 \end{cases} \] (D.51)

**D.3.2.3 van Leer Preconditioner**

Name: VLDA [138]
Solution vector: \( dQ || = (a dp/(\sqrt{\gamma} \rho), dq, dr, ds, a dT/(\sqrt{\gamma(\gamma - 1)}T)) \)

\[
\Gamma_{VLDA}^{-1} = \begin{bmatrix}
\beta & 0 & 0 & 0 & 0 \\
0 & 1 & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 0 \\
0 & 0 & 0 & 1 & 0 \\
0 & 0 & 0 & 0 & 1
\end{bmatrix} \] (D.52)

Preconditioning parameters:
\[
\beta = \begin{cases} 
M^2, & \text{Re} \gg 1 \\
\frac{M^2}{\text{Re}}, & \text{Re} \ll 1, \frac{M^2}{\text{Re}} \ll 1 \\
\frac{1}{\text{Re}}, & \text{Re} \ll 1, \frac{M^2}{\text{Re}} \gg 1
\end{cases}
\] (D.53)

### D.3.3 Time Derivative Viscous Preconditioners

#### D.3.3.1 Merkle Preconditioner

**Choi Merkle Viscous**

Name: CMv [153, 157, 159, 167]

Solution vector: \( Q_v = (p, u, v, w, T)^T \)

\[
\Gamma_{CMv} = \begin{bmatrix} 
\frac{1}{\beta M^2} & 0 & 0 & 0 & 0 \\
\frac{u}{\beta M^2} & \rho & 0 & 0 & 0 \\
\frac{v}{\beta M^2} & 0 & \rho & 0 & 0 \\
\frac{w}{\beta M^2} & 0 & 0 & \rho & 0 \\
\frac{\rho + p}{\rho \beta M^2} - \delta & \rho u & \rho v & \rho w & \frac{\rho R}{\gamma - 1}
\end{bmatrix}
\] (D.54)

\( \delta = 1 \) is the 1991 version of CMv preconditioner. This preconditioner is similar to VM (See Appendix D.3.2.2). Extensions have been made to this preconditioner by Colin *et al.* [159].

Preconditioning parameters [153, 157]:

\[
\delta = 1 \\
\beta = k \gamma RT \\
k = \max \left[ 1, \frac{\alpha (\alpha - 1)}{M^2 (\alpha - 1 + \frac{\gamma RT}{\alpha})} \right] \\
\alpha = \frac{\text{CFL}}{\text{VNN}} \text{Re}_{\Delta x}
\]

**CMv-Buelow**

Name: CMv-B [103, 169]

231
Solution vector: \( \mathbf{Q}_v = (p, u, v, w, T)^T \)

\[
\Gamma_{CMv-B} = \begin{bmatrix}
\frac{1}{\varepsilon c^2} & 0 & 0 & 0 & 0 \\
\frac{u}{\varepsilon c^2} & \rho & 0 & 0 & 0 \\
\frac{v}{\varepsilon c^2} & 0 & \rho & 0 & 0 \\
\frac{w}{\varepsilon c^2} & 0 & 0 & \rho & 0 \\
\frac{h + \frac{q^2}{2}}{\varepsilon c^2} & -\delta & \rho u & \rho v & \rho w & \rho C_p
\end{bmatrix}
\]  

(D.55)

\( \varepsilon \) is used to activate the inviscid / viscous preconditioner,

\[
\varepsilon = \max(\varepsilon_{inv}, \varepsilon_{vis})
\]  

(D.56)

### D.3.3.2 Weiss/Smith Preconditioner

Name: WS [90]

Solution vector: \( \mathbf{Q} = (\rho, \rho u, \rho v, \rho w, E_t)^T \)

Preconditioning parameters: \( \beta \)

\[
\Gamma_{WS} = \begin{bmatrix}
\beta & 0 & 0 & 0 & \rho_T \\
\beta u & \rho & 0 & 0 & \rho_T u \\
\beta v & 0 & \rho & 0 & \rho_T v \\
\beta w & 0 & 0 & \rho & \rho_T w \\
\beta h_0 - 1 & \rho u & \rho v & \rho w & \rho_T h_0 + \rho C_p
\end{bmatrix}
\]  

(D.57)

where \( \rho_T = \frac{\partial \rho}{\partial T}|_p \), \( \rho_p = \frac{\partial \rho}{\partial p}|_T \), \( C_p \) is the specific heat of constant pressure,

\[
\beta = \left( \frac{1}{U_r} - \frac{\rho_T}{\rho C_p} \right)
\]  

(D.58)

where the reference velocity is defined for an ideal gas is,

\[
U_{r \text{ gas}} = \begin{cases} 
\varepsilon c, & |q| < \varepsilon c \\
|m|, & \varepsilon c < |q| < c \\
c, & |q| > c
\end{cases}
\]

where \( U_{max} \) is the maximum velocity in the flow field and \( \varepsilon \) is a small number \((\sim 10^{-5})\). \( U_r \) is also limited so that it does not exceed the local diffusion velocity, \( U_r = \max(U_r, \nu/\Delta x) \).
D.3.4 Link Between CMi / CMv / WS Preconditioners

Name: CMi/CMv/WS [115]
Solution vector: \( dQ_{||} = (\frac{dp}{pc}, dq, dr, dp - c^2d\rho)^T \)

\[
\Gamma_{WS} = \begin{bmatrix}
\beta & 0 & 0 & 0 & -\delta_{cmv}\beta \\
0 & 1 & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 0 \\
0 & 0 & 0 & 1 & 0 \\
\delta_{cmi}(\beta - 1) & 0 & 0 & 0 & 1 \\
\end{bmatrix}
\] (D.59)

This preconditioner links the CMi, CMv, and WS preconditioner when the following parameters are defined,

\[
CMi : \quad \delta_{cmi} = 1, \delta_{cmv} = 0 \\
CMv : \quad \delta_{cmi} = 0, \delta_{cmv} = 1 \\
WS : \quad \delta_{cmi} = 0, \delta_{cmv} = 0
\]

D.3.5 Pletcher/Chen Preconditioner

Name: PC [171,172]
Solution vector: \( Q_v = (p, u, v, T)^T \)
Preconditioning parameters: None

\[
\Gamma_{PC} = \begin{bmatrix}
\frac{1}{T} & 0 & 0 & -\frac{p}{RT^2} \\
\frac{\nu}{T} & \frac{p}{RT} & 0 & -\frac{pu}{RT^2} \\
\frac{\nu}{T} & 0 & \frac{p}{RT} & -\frac{pv}{RT^2} \\
\frac{1}{\gamma} + \frac{1}{2}\frac{q^2}{C_pT} & \frac{pu}{RC_pT} & \frac{pv}{RC_pT} & \frac{pq^2}{2C_pRT^2} \\
\end{bmatrix}
\] (D.60)
D.3.6 Asymptotic Preconditioners

D.3.6.1 Merkle 1988

Name: M88 [119]
Solution vector: \( Q = (p_1/\beta, u, v, T)^T \)

\[
\Gamma_{M88} \frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} = 0
\]  
(D.61)

\[
\Gamma_{M88} = \begin{bmatrix}
\frac{\rho u}{p_0} & 0 & 0 \\
\frac{\rho u}{p_0} & \rho & 0 \\
\frac{\rho v}{p_0} & 0 & \rho \\
1 & 0 & 0 & \rho
\end{bmatrix}
\]  
(D.62)

with the flux vectors defined as,

\[
E = \left\{ \begin{array}{c}
\rho u \\
\rho u^2 + p_1 \\
\rho u v \\
(pE_t + p) u
\end{array} \right\}, \\
F = \left\{ \begin{array}{c}
\rho v \\
\rho uv \\
\rho v^2 + p \\
(pE_t + p_1) v
\end{array} \right\}
\]

Preconditioning parameters:

\[
\beta = \max \left\{ 1, \frac{u^2}{4T} \left[ \left( \frac{2 \text{CFL}}{\text{VNN Re}_{\Delta x}} - 1 \right)^2 - 1 \right] \right\}
\]  
(D.63)

D.3.6.2 Merkle Asymptotic

Name: Masymp [92, 108, 176, 177]
Solution vector: \( Q_v = (p, u, v, w, T)^T \)
\[
\Gamma_{\text{Masym}} = \begin{bmatrix}
\rho_p' & 0 & 0 & 0 & \rho_T' \\
 u\rho_p' & \rho & 0 & 0 & u\rho_T' \\
v\rho_p' & 0 & \rho & 0 & v\rho_T' \\
w\rho_p' & 0 & 0 & \rho & w\rho_T' \\
h_0\rho_p' - (1 - \rho h_p) & \rho u & \rho v & \rho w & h_0\rho_T' + \rho h_T
\end{bmatrix}
\] (D.64)

Where \( h_0 \) is the total enthalpy, \( k_T = 0 \) for a perfect gas and \( k_T = 1 \) for an arbitrary equation of state and,

\[
h_T = \frac{\partial h}{\partial T}
\bigg|_p, \quad \rho_p = \frac{\partial \rho}{\partial p}
\bigg|_T, \quad \rho_T = \frac{\partial \rho}{\partial T}
\bigg|_p
\] (D.65)

**Preconditioning parameters Set 1** [92, 144, 177]:

\[
\rho_p' = \min \left[ \max \left( \frac{k_i}{V_{\text{inv}}^2}, \frac{k_v}{V_{\text{vis}}^2/Re_{\Delta x}^2}, \frac{k_u}{\text{unstdy}^2 St^2} \right), \rho_p \right]
\]

\[
\rho_T' = \min(1, \frac{\text{Re}_{\Delta x}}{Pr}) k_T \rho_T
\]

\[
k_i \approx k_v \approx k_u \approx 1
\] (D.66)

with \( St \) the Strouhal number and,

\[
V_{\text{inv}} \approx q, \quad V_{\text{vis}} \approx V_{\text{inv}}
\]

\[
V_{\text{unstdy}} \approx \max \left[ \frac{L_x}{\pi \Delta t}, \frac{L_y}{\pi \Delta t} \right]
\] (D.67)

Where \( L \) is a characteristic length scale of the flow (e.g. cavity length in a cavity flow).

**Preconditioning parameters Set 2** [108]
\[ \rho_p' = \frac{1}{V_p^2} - \frac{\rho_T'(1 - \rho h_p)}{\rho h_T} \]
\[ \rho_T' = \delta \rho_T \]

with

\[ V_p = \min \left[ \max \left( V_{\text{inv}}, V_{\text{pgr}}, V_{\text{vis}}, V_{\text{unstdy}} \right), c \right] \]
\[ V_{\text{inv}} = \frac{q}{Re_{\Delta x}} \]
\[ V_{\text{vis}} = \frac{\sqrt{|\Delta p|}}{\rho} \]
\[ V_{\text{pgr}} = \frac{L_x}{\pi \Delta t}, \frac{L_y}{\pi \Delta t} \]  

(D.68)

\section*{D.3.7 Hybrid Preconditioner}

In some fluid dynamic problems there are pockets of very low speed flow that occur in the midst of moderate to high subsonic flow, e.g. stalled airfoils, converging nozzles, etc. The convergence rate can be dramatically increased through preconditioning, but most preconditioners are not effective for high subsonic Mach numbers except for VLR91, which is good for all Mach numbers. For preconditioners that perform better than the VLR91 at low Mach numbers (e.g. the Modified Turkel) a hybrid preconditioner can be used to link the two preconditioners over a range of Mach numbers [110]. Using a linear interpolation the preconditioner is,

\[ \Gamma(M) = \sigma \Gamma_1(M_{\text{beg}}) + (1 - \sigma) \Gamma_2(M_{\text{end}}) \]  

(D.69)

where \( M_{\text{beg}} \) and \( M_{\text{end}} \) is the Mach number range over which to interpolate, \( \sigma = (M - M_{\text{end}})/(M_{\text{beg}} - M_{\text{end}}) \), \( \Gamma_1 \) is the low Mach number preconditioner, and \( \Gamma_2 \) is the high Mach number preconditioner.
Bibliography


238


246


[151] **Fiedler, J.** and **F. DiMare** (2012), “Generalised implementation for arbitrary three-dimensional geometries,” Vienna, Austria, ECCOMAS.


Vita

Brent S. Paul

Brent S. Paul was born in Camp Hill, Pennsylvania in 1972 to parents Joanne and Glenn Paul. He has a younger brother, Brad. He married his wife Roberta in 2003 and they have two children, Brodie and Jenna.

He attended The Pennsylvania State University and received his B.S. degree in Aerospace Engineering in 1994. He continued his studies and received his M.S degree in 1996. His Master’s thesis focused on the development of a numerical boundary condition for the simulation of flow-acoustic properties of jet engine liners - “Time-dependent Simulations of a Frequency-dependent Impedance Boundary Condition in Computational Aeroacoustics”. This work was completed under the guidance of Dr. Phil Morris.

In 1997 Brent started work at the Electric Boat Corporation in Groton, CT. He became a Senior Engineer and was responsible for the analytical and numerical predictions of various hydroacoustic phenomena including external flow noise and propeller noise. It was during his time at Electric Boat that he started his doctoral work in acoustics. In 2004 he moved back to Pennsylvania and supported the Acoustic Department at Anteon Corporation, where he continued research and engineering in support of the U.S. Navy. He is currently a Principal Engineer at Alion Science and Technology Corporation (who purchased Anteon in 2011) using his background and experience in computational fluid dynamics and acoustics to develop engineering solutions. He is a member of the American Society of Mechanical Engineers where he has served on the Executive and Technical Committees of the Noise Control and Acoustic Division.