Model Reduction of Computational Aerothermodynamics for Multi-Discipline Analysis in High Speed Flows

Dissertation

Presented in Partial Fulfillment of the Requirements for the Degree Doctor of Philosophy in the Graduate School of The Ohio State University

By

Andrew Rippetoe Crowell, M.S.

Graduate Program in Aeronautical and Astronautical Engineering

The Ohio State University

2013

Dissertation Committee:
Jack J. McNamara, Advisor
Jeffrey Bons
Thomas Eason III
Mo-How Herman Shen
Mei Zhuang
Abstract

This dissertation describes model reduction techniques for the computation of aerodynamic heat flux and pressure loads for multi-disciplinary analysis of hypersonic vehicles.

NASA and the Department of Defense have expressed renewed interest in the development of responsive, reusable hypersonic cruise vehicles capable of sustained high-speed flight and access to space. However, an extensive set of technical challenges have obstructed the development of such vehicles. These technical challenges are partially due to both the inability to accurately test scaled vehicles in wind tunnels and to the time intensive nature of high-fidelity computational modeling, particularly for the fluid using Computational Fluid Dynamics (CFD).

The aim of this dissertation is to develop efficient and accurate models for the aerodynamic heat flux and pressure loads to replace the need for computationally expensive, high-fidelity CFD during coupled analysis. Furthermore, aerodynamic heating and pressure loads are systematically evaluated for a number of different operating conditions, including: simple two-dimensional flow over flat surfaces up to three-dimensional flows over deformed surfaces with shock-shock interac-
tion and shock-boundary layer interaction. An additional focus of this dissertation is on the implementation and computation of results using the developed aerodynamic heating and pressure models in complex fluid-thermal-structural simulations.

Model reduction is achieved using a two-pronged approach. One prong focuses on developing analytical corrections to isothermal, steady-state CFD flow solutions in order to capture flow effects associated with transient spatially-varying surface temperatures and surface pressures (e.g., surface deformation, surface vibration, shock impingements, etc.). The second prong is focused on minimizing the computational expense of computing the steady-state CFD solutions by developing an efficient surrogate CFD model.

The developed two-pronged approach is found to exhibit balanced performance in terms of accuracy and computational expense, relative to several existing approaches. This approach enables CFD-based loads to be implemented into long duration fluid-thermal-structural simulations.
Dedication

For my beautiful wife Sarah and our lovely daughter Evie.
Acknowledgments

I would like to thank my advisor Prof. Jack McNamara, who encouraged me to pursue graduate school and ultimately is responsible for my funding throughout graduate school. Professor McNamara’s guidance and unrelenting attitude to always dig deeper into a problem in order to understand the fundamental physics led to many of the novel contributions contained in this dissertation, and while it was not always easy, I am truly grateful. I would also like to thank Profs. Mei Zhuang, Mo-How Herman Shen, and Jeffrey Bons for serving on my dissertation and candidacy committees, and Dr. Thomas Eason of the AFRL SSC for serving on my dissertation committee.

I gratefully acknowledge the support which enabled this work. This research was performed with government support awarded by AFOSR Grant FA9550-11-1-0036, with Dr. John Schmisseur as Program Manager; NASA award NNX08AB32A with Mr. Don Soloway as technical lead and Dr. Jorge Bardina as technical monitor; the DoD Science Mathematics and Research for Transformation (SMART) Scholarship, and in part by an Air Force Summer Faculty Fellowship awarded to Prof. McNamara by the U.S. Air Force Research Laboratory, Structural Sciences
Center. Additionally, computational resources were provided in part by an allocation of computing time from the Ohio Supercomputer Center.

Contributions to this work have been made by researchers at the U.S. Air Force Research Laboratory and by former and current graduate students. I gratefully acknowledge the technical insights of Drs. Ravi Chona, Tom Eason, and S. Michael Spottswood of the AFRL Structural Sciences Center. Additionally, Dr. Adam Culler and Brent Miller enabled significant portions of this research. I would like to thank Dr. Culler for providing his thermo-elastic hypersonic vehicle panel model and his results. I would also like to thank Brent Miller for enabling the spatial integration of pressure inside of Dr. Culler’s thermo-elastic panel model, and for the thermal and structural models as well as the time marching scheme for the fluid-thermal-structural shock-impingement panel model.

I am also grateful for all of the fellow graduate students in my office. To Dr. Krista Kecskemety, Brent Miller, Steve Nogar, Zach Riley, Rohit Deshmukh, and Zach Witeof, you have all become great friends and made my time in the office much more enjoyable. To my wife, Sarah Crowell, you have been so patient and loving throughout this whole process, I am so blessed to be married to such a wonderful person. I would also like to thank my mother, Katie Rader, for all of the love and support she has shown me; all while raising three boys by herself. You are an amazing woman and I am thankful that you are my mother. Finally, I would like to thank God for the many blessings which fill my life so completely.
Vita

2008 ................. B.S. Aeronautical and Astronautical Engineering, The Ohio State University

2009 ................. M.S. Aeronautical and Astronautical Engineering, The Ohio State University

2008 - 2010 ............ Graduate Research Assistant, The Ohio State University

2010 - 2013 ............ SMART Scholarship, The Ohio State University
Publications

Journal Publications


Conference Publications


Fields of Study

Major Field: Aeronautical and Astronautical Engineering
# Table of Contents

Abstract .................................................................................................................. ii

Dedication ................................................................................................................ iv

Acknowledgments .................................................................................................. v

Vita .......................................................................................................................... vii

List of Tables ........................................................................................................... xx

List of Figures ......................................................................................................... xxvi

List of Symbols ....................................................................................................... xxxix

Chapter

1 Introduction and Objectives .................................................................................. 1

1.1 Introduction ..................................................................................................... 1

1.2 Literature Review .......................................................................................... 7

1.2.1 Fluid-Thermal-Structural Interactions ..................................................... 7

1.2.2 Aerodynamic Heating ........................................................................... 10

xiii
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.2.3</td>
<td>Aerodynamic Pressure</td>
<td>14</td>
</tr>
<tr>
<td>1.2.4</td>
<td>Computational Model Reduction</td>
<td>16</td>
</tr>
<tr>
<td>1.3</td>
<td>Objectives of this Dissertation</td>
<td>19</td>
</tr>
<tr>
<td>1.4</td>
<td>Key Novel Contributions in this Dissertation</td>
<td>21</td>
</tr>
<tr>
<td>2</td>
<td>Computational Configurations</td>
<td>22</td>
</tr>
<tr>
<td>2.1</td>
<td>Two-Dimensional Generic Panel</td>
<td>23</td>
</tr>
<tr>
<td>2.2</td>
<td>Two-Dimensional HSV Panel</td>
<td>26</td>
</tr>
<tr>
<td>2.3</td>
<td>Two-Dimensional Shock Impingement Panel</td>
<td>30</td>
</tr>
<tr>
<td>2.4</td>
<td>Three-Dimensional Shock Impingement Panel</td>
<td>35</td>
</tr>
<tr>
<td>2.5</td>
<td>Three-Dimensional Control Surface</td>
<td>40</td>
</tr>
<tr>
<td>3</td>
<td>CFD Model Reduction</td>
<td>44</td>
</tr>
<tr>
<td>3.1</td>
<td>Model Reduction Framework</td>
<td>45</td>
</tr>
<tr>
<td>3.2</td>
<td>Computational Model Reduction Techniques</td>
<td>49</td>
</tr>
<tr>
<td>3.2.1</td>
<td>Kriging</td>
<td>49</td>
</tr>
<tr>
<td>3.2.2</td>
<td>Proper Orthogonal Decomposition</td>
<td>52</td>
</tr>
<tr>
<td>3.3</td>
<td>Comparison of Methods</td>
<td>55</td>
</tr>
<tr>
<td>3.3.1</td>
<td>Steady-State Aerodynamic Heating</td>
<td>57</td>
</tr>
<tr>
<td>3.3.2</td>
<td>Steady-State Aerodynamic Pressure</td>
<td>57</td>
</tr>
<tr>
<td>3.3.3</td>
<td>Computational Expense</td>
<td>60</td>
</tr>
<tr>
<td>4</td>
<td>Aerodynamic Heating</td>
<td>62</td>
</tr>
<tr>
<td>4.1</td>
<td>Turbulence and Analytical Model Analysis</td>
<td>62</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
<td></td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td></td>
</tr>
<tr>
<td>4.2 Coupling Approaches for Surface Temperature and Aerodynamic Heating</td>
<td>66</td>
<td></td>
</tr>
<tr>
<td>4.2.1 Linearization Method</td>
<td>67</td>
<td></td>
</tr>
<tr>
<td>4.2.2 Integral Methods for Aerodynamic Heating</td>
<td>68</td>
<td></td>
</tr>
<tr>
<td>4.2.3 Pointwise Approximation</td>
<td>70</td>
<td></td>
</tr>
<tr>
<td>4.2.4 Parameterization of Surface Temperature Using Spatial Basis Functions</td>
<td>71</td>
<td></td>
</tr>
<tr>
<td>4.2.5 Corrected Pointwise Approach</td>
<td>72</td>
<td></td>
</tr>
<tr>
<td>4.3 Comparison of the Surface Temperature Coupling Strategies</td>
<td>86</td>
<td></td>
</tr>
<tr>
<td>4.3.1 Construction and Verification of CFD Surrogates for Flat Surfaces</td>
<td>87</td>
<td></td>
</tr>
<tr>
<td>4.3.2 Comparison of Aerodynamic Heating Models for Laminar Flow over a Flat Surface</td>
<td>94</td>
<td></td>
</tr>
<tr>
<td>4.3.3 Comparison of Aerodynamic Heating Models for Turbulent Flow over a Flat Surface</td>
<td>98</td>
<td></td>
</tr>
<tr>
<td>4.3.4 Construction and Verification of CFD Surrogates with Deformation</td>
<td>99</td>
<td></td>
</tr>
<tr>
<td>4.3.5 Comparison of Aerodynamic Heating Models with Deformation</td>
<td>102</td>
<td></td>
</tr>
<tr>
<td>5 Aerodynamic Pressure</td>
<td>105</td>
<td></td>
</tr>
<tr>
<td>5.1 Turbulence and Analytical Model Analysis</td>
<td>106</td>
<td></td>
</tr>
</tbody>
</table>
6.7 Nonlinear Post-Flutter Response ........................................... 170

7 Fluid-Thermal-Structural Analysis: Shock Impingement Panel ........ 177
  7.1 Aerodynamic Heat Flux Modeling ........................................ 177
      7.1.1 Construction and Verification of CFD Surrogates ............... 178
      7.1.2 Comparison of Aerodynamic Heating Models ..................... 181
  7.2 Unsteady Aerodynamic Pressure Modeling .............................. 184
      7.2.1 Stationary Shock ................................................. 184
      7.2.2 Oscillating Shock ................................................ 195
  7.3 Fluid-Thermal-Structural Method of Solution ........................... 201
  7.4 Fluid-Thermal Analysis ................................................ 204
  7.5 Fluid-Thermal-Structural Analysis ...................................... 210
      7.5.1 Effect of Backpressure on Buckling Direction ................. 211
      7.5.2 Fluid-Thermal-Structural Response ............................... 213
  7.6 Three-Dimensional Aerodynamic Heat Flux Modeling .................. 219
      7.6.1 Neglecting Deformation and Shock Impingement ................ 219
      7.6.2 Including Deformation .......................................... 225
      7.6.3 Including Deformation and Shock Impingement .................. 230
  7.7 Three-Dimensional Unsteady Pressure Modeling with Shock Impingement ........................................... 236

8 Computational Requirements .............................................. 242
  8.1 Two-Dimensional Fluid-Thermal Structural Simulation .............. 242
8.1.1 HSV Panel ........................................ 243
8.1.2 Two-Dimensional Shock Impingement Panel ............ 245
8.2 Three-Dimensional Fluid-Thermal Structural Simulation .... 247
  8.2.1 Three-Dimensional Shock Impingement Panel .......... 248
  8.2.2 Control Surface .................................. 249

9 Concluding Remarks ........................................ 252
  9.1 Principal Conclusions Obtained in this Study .......... 252
  9.2 Recommendations for Future Research ..................... 257

Bibliography .................................................. 261

Appendix ....................................................... 273
  10.1 The CFL3D Euler/Navier-Stokes Solver ................. 273
    10.1.1 CFL3D Fluid Equations .......................... 273
    10.1.2 CFL3D Unsteady Mesh Deformation ................. 274
  10.2 Generic Panel Mesh Convergence ........................ 275
  10.3 HSV Panel Mesh Convergence ............................ 278
  10.4 Two-Dimensional Shock Impingement Panel Mesh Convergence . 282
  10.5 Three-Dimensional Shock Impingement Panel Mesh Convergence 289
  10.6 Control Surface Mesh Convergence ....................... 293
  10.7 Eckert’s Reference Methods ............................ 297
  10.8 Chapman and Rubesin’s Method ........................ 299
    10.8.1 Laminar Boundary Layer Equations ................. 299
10.8.2 Particular Solution ............................................. 301
10.8.3 Homogenous Solution .......................................... 302
10.8.4 Complete Solution ............................................... 302
10.8.5 Heat Flux .......................................................... 303
## List of Tables

<table>
<thead>
<tr>
<th></th>
<th>Table Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Generic panel parameter space.</td>
<td>25</td>
</tr>
<tr>
<td>2.2</td>
<td>HSV panel geometry.</td>
<td>28</td>
</tr>
<tr>
<td>2.3</td>
<td>HSV panel parameter space.</td>
<td>29</td>
</tr>
<tr>
<td>2.4</td>
<td>2-D shock impingement panel and shock generator geometry</td>
<td>32</td>
</tr>
<tr>
<td>2.5</td>
<td>2-D shock impingement panel parameter space.</td>
<td>34</td>
</tr>
<tr>
<td>2.6</td>
<td>3-D shock impingement panel and shock generator geometry</td>
<td>37</td>
</tr>
<tr>
<td>2.7</td>
<td>3-D shock impingement panel parameter space.</td>
<td>39</td>
</tr>
<tr>
<td>2.8</td>
<td>Control surface parameter space.</td>
<td>42</td>
</tr>
<tr>
<td>2.9</td>
<td>Comparison of the Lockheed F-104 Starfighter wing to the control surface model [30]</td>
<td>43</td>
</tr>
<tr>
<td>3.1</td>
<td>Comparison of wall times for kriging, POD, and CFD.</td>
<td>60</td>
</tr>
<tr>
<td>4.1</td>
<td>Percent difference in aerodynamic heating predictions using different modeling</td>
<td>65</td>
</tr>
<tr>
<td></td>
<td>approaches for the HSV panel.</td>
<td></td>
</tr>
<tr>
<td>4.2</td>
<td>Numerical Values of $Y_n'(0)$ [67].</td>
<td>74</td>
</tr>
<tr>
<td>4.3</td>
<td>Approximated exponents and coefficients for Eq. (4.30) for laminar and turbulent</td>
<td>85</td>
</tr>
<tr>
<td></td>
<td>flow.</td>
<td></td>
</tr>
</tbody>
</table>
6.2 Error in GAFs for oscillations of the HSV panel for several methods compared to the Euler CFD solution (EU) over all three modes.

6.3 Difference in GAFs for the HSV panel for the Navier-Stokes solution with 300 K surface temperature compared to several other methods over all three modes.

6.4 Error of several surrogate GAFs for the HSV panel compared to the Navier-Stokes solution with 300 K surface temperature over all three modes.

6.5 Properties of the thermal structure of the HSV panel at 300 K [21].

6.6 Aerodynamic heating and pressure sub-models for the fluid-thermal-structural response of the HSV panel.

6.7 Parameters used to compute the fluid-thermal-structural response of the HSV panel.

6.8 Percent difference in the flight times to the onset of flutter predicted using the “CIM”, “Linear”, and analytical models used by Culler and McNamara [21] (“C–M”).

6.9 Percent difference between the “CIM” and “Linear” models from the “C–M” model for pressure, heating, panel displacement, and mid-plate temperature rise from 300 K for $M_1 = 7.5$ at 1500 sec.

6.10 Percent difference the “CIM” and “Linear” models from the “C–M” model for pressure, heating, panel displacement, and mid-plate temperature rise from 300 K for $M_1 = 10.5$ at 40 sec.
7.1 Error in the aerodynamic heat flux models for the 2-D deformable panel subject to shock impingement compared to CFD. 181

7.2 Error in GAFs for a stationary shock impinging on a 2-D panel, comparing analytical and Euler-based methods for oscillations of the first three panel modes. 185

7.3 Error in GAFs for a stationary shock impinging on a 2-D panel, comparing Navier-Stokes, analytical, and Euler-based methods for oscillations of the first three panel modes. 190

7.4 Error in GAFs for a stationary shock impinging on a 2-D panel, comparing Navier-Stokes, and surrogate-based methods for oscillations of the first three panel modes. 192

7.5 Error in GAFs for an oscillating shock impinging on a vibrating 2-D panel, comparing Navier-Stokes, analytical, and Euler based methods for oscillations of the first panel mode. 195

7.6 Error in GAFs for an oscillating shock impinging on a vibrating 2-D panel, comparing Navier-Stokes and surrogate-based methods for oscillations of the first panel mode. 199

7.7 Heat conduction properties of the 2-D shock impingement panel at 220K. 202

7.8 Numerical properties of thermal and structural models of the 2-D shock impingement panel. 203
7.9 Aerodynamic heating and pressure sub-models for the fluid-thermal-structural response of the HSV panel.

7.10 Percent differences in the spatially averaged through-thickness temperature rise and generalized aerodynamic forces for several cases compared to the unsteady case for 2.0 seconds of response.

7.11 Descriptions of the fluid-thermal-structural cases for the 2-D shock impingement panel.

7.12 Comparison of the critical backpressure which causes the panel to switch from buckling out of the flow to buckling into the flow ($P_\infty = 2970$Pa).

7.13 Error in the aerodynamic heat flux for several models for the 3-D panel without deformation or shock impingement compared to CFD.

7.14 Error in the aerodynamic heat flux for several models for the 3-D panel without shock impingement compared to CFD.

7.15 Error in the aerodynamic heat flux for several models for the 3-D deformable panel subject to shock impingement compared to CFD.

7.16 Error in GAFs for an oscillating shock impinging on a vibrating 3-D panel, comparing Navier-Stokes, analytical, and Euler based methods for oscillations of the first panel mode.
7.17 Error in GAFs for an oscillating shock impinging on a vibrating 3-D panel, comparing Navier-Stokes and surrogate-based methods for oscillations of the first panel mode. ........................................ 241

8.1 Computational cost of different fluid-thermal-structural modeling approaches for the HSV panel. ........................................ 244

8.2 Computational cost of different fluid-thermal-structural modeling approaches for the 2-D panel with shock impingement. .............. 246

8.3 Computational cost of different fluid-thermal-structural modeling approaches for the 3-D panel with shock impingement. .............. 249

8.4 Computational cost of different fluid-thermal-structural modeling approaches for the control surface. ........................................ 250

10.1 HSV panel unsteady CFD parameters. ...................................... 279

10.2 2-D shock impingement panel unsteady CFD parameters for the stationary shock generator cases. ...................................... 288

10.3 2-D shock impingement panel unsteady CFD parameters for the oscillating shock generator cases ...................................... 288

10.4 3-D shock impingement panel unsteady CFD parameters. ........... 290

10.5 Control surface unsteady CFD parameters. ................................. 293
# List of Figures

1.1 3 view of the NASA X-43A Hyper-X hypersonic test vehicle [12]  
1.2 Fluid-Thermal-Structural Coupling  
2.1 Generic panel computational mesh  
2.2 HSV panel mesh  
2.3 Panel located on an inclined surface of a wedge-shaped forebody  
2.4 Schematic of the 2-D shock impingement panel and tunnel configuration with a shock-generating wedge in supersonic flow. *(SSI: Shock-Shock Interactions, STBLI: Shock-Turbulent Boundary Layer Interactions)*  
2.5 Schematic of the geometry of the 2-D shock impingement panel and tunnel  
2.6 2-D shock impingement panel mesh  
2.7 3-D shock impingement panel mesh  
2.8 3-D shock impingement panel mesh (x-z view)  
2.9 3-D shock impingement panel mesh (y-z view)  
2.10 Control surface geometry [30]
2.11 Control surface mesh. .......................................................... 41
2.12 First two free vibration modes of the control surface [30]. ....... 43
3.1 Schematic of process for modeling steady-state CFD data. ......... 47
3.2 Mean and maximum percent errors for kriging and POD-based steady-state heat flux models. ........................................... 58
3.3 Mean and maximum errors for kriging and POD-based steady-state pressure models. ....................................................... 59
4.1 Representative temperature profile used to compare several analytical and turbulence model aerodynamic heating predictions for the HSV panel. ......................................................... 63
4.2 Comparison of aerodynamic heating predictions for several HSV panel shapes, $M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa$. ............. 64
4.3 Temperature profile. ............................................................... 79
4.4 Correction ($\Delta q$) dependence on boundary layer edge density ($C_1$) for laminar and turbulent flow. $\rho_\infty = 10.0 - 375.0 \, g/m^3, M_\infty = 3.0, T_\infty = 216K$. ......................... 80
4.5 Correction ($\Delta q$) dependence on boundary layer edge temperature ($C_2$) for laminar and turbulent flow. $T_\infty = 210 - 340 \, K, U_\infty = 1,180 \, m/s, \rho_\infty = 46.94 \, g/m^3$. ............................. 82
4.6 Correction ($\Delta q$) dependence on boundary layer edge velocity ($C_3$) for laminar and turbulent flow. $a_\infty = 295.0 \, m/s, M_\infty = 2.5 - 5.75, \rho_\infty = 46.94 \, g/m^3, T_\infty = 216 \, K$ .......................... 83

xxvii
4.7 Mid-chord aerodynamic heating for laminar flow for the generic panel, for the “Linear” heat flux model and the pointwise model (“Iso”). Alt. = 24km. ........................................ 90

4.8 Mid-chord aerodynamic heating for turbulent flow for the generic panel, for the “Linear” heat flux model and the pointwise model (“Iso”). Alt. = 24km. ........................................ 91

4.9 Comparison of several laminar aerodynamic heating predictions to the benchmark (CFD) for the flat generic panel: \( M_{\infty} = 5.181 \), \( \rho_{\infty} = 196.8 \text{ g/m}^3 \), and \( T_{\infty} = 325.3 \text{ K} \). ........................................ 96

4.10 Comparison of Chapman-Rubesin models with \( n = 2, 4, \) and \( 8 \) for the flat generic panel: \( M_{\infty} = 5.007 \), \( \rho_{\infty} = 351.6 \text{ g/m}^3 \), and \( T_{\infty} = 244.5 \text{ K} \). ........................................ 98

4.11 Comparison of several turbulent aerodynamic heating predictions to the Benchmark (CFD) for the flat generic panel: \( M_{\infty} = 3.278 \), \( \rho_{\infty} = 334.1 \text{ g/m}^3 \), \( T_{\infty} = 314.5 \text{ K} \). ........................................ 100

4.12 Turbulent aerodynamic heat flux comparison of several models compared to the Benchmark (CFD) for the generic panel with deformation. For all of the cases: \( M_{\infty} = 3.0 \), Alt. = 24km. ........................................ 104

5.1 Direction of the free stream relative to a 3-D structure. .................. 109

5.2 Representative temperature profile used to compare several analytical and turbulence model aerodynamic pressure predictions for the HSV panel. .................. 111
5.3 Comparison of aerodynamic pressure predictions for several HSV panel shapes, $M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa$.

5.4 Euler, steady-state Euler, and piston theory Generalized Aerodynamic Forces (GAFs) for oscillations of the control surface. $f = 1, 25, 50, 100, 150, 200 Hz, M_\infty = 8.0, Alt. = 40 km$.

5.5 Unsteady components of the GAFs for the Euler and Euler with local piston theory for oscillations of the control surface. $a_1^* = 1.0, a_2^* = 0.5, f = 1, 25, 50, 100, 150, 200 Hz, M_\infty = 8.0, Alt. = 40 km$.

5.6 Navier-Stokes and Euler Generalized Aerodynamic Forces (GAFs) for surface temperatures of free stream (262 K) and 1200 K for oscillations of the control surface. $f = 1, 25, 50, 100, 150, 200 Hz, M_\infty = 8.0, Alt. = 40 km$.

5.7 Unsteady components of the GAFs for Navier-Stokes, piston theory, and local piston theory for oscillations of the control surface. $a_1^* = 1.0, a_2^* = 0.5, f = 1, 25, 50, 100, 150, 200 Hz, M_\infty = 8.0, Alt. = 40 km$.

5.8 Navier-Stokes and surrogate Generalized Aerodynamic Forces (GAFs) for the control surface. $f = 1, 25, 50, 100, 150, 200 Hz, M_\infty = 8.0, Alt. = 40 km$.

5.9 Navier-Stokes, Euler, piston theory, and surrogate GAFs for the control surface. $T_w = 1200 K, Alt. = 40 km$.
6.1 Turbulent aerodynamic heat flux comparison of several models compared to the Benchmark (CFD) for the HSV panel. For this case:

\[ M_2 = 8.36, \rho_2 = 39.35 \text{ g/m}^3, T_2 = 324.4 \text{ K}. \]

6.2 Euler and piston theory Generalized Aerodynamic Forces (GAFs) for oscillations of the HSV panel. \( f = 140 \text{ Hz}, M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa. \)

6.3 Euler and piston theory corrected steady-state Euler model GAFs for oscillations of the HSV panel. \( f = 140 \text{ Hz}, M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa. \)

6.4 Euler and local piston theory corrected steady-state Euler model GAFs for oscillations of the HSV panel. \( f = 140 \text{ Hz}, M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa. \)

6.5 Navier-Stokes and Euler Generalized Aerodynamic Forces (GAFs) for surface temperatures of 300K and 900K for oscillations of the HSV panel. \( f = 140 \text{ Hz}, M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa. \)

6.6 Navier-Stokes, Euler, and corrected steady-state Navier-Stokes Generalized Aerodynamic Forces (GAFs) for oscillations of the HSV panel. \( f = 140 \text{ Hz}, M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa, T_w = 300K, T_r = 0.085. \)

6.7 Navier-Stokes and the corrected steady-state pointwise surrogate Generalized Aerodynamic Forces (GAFs) for the HSV panel: \( f = 140 \text{ Hz}, M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa, T_w = 300K, T_r = 0.085. \)
6.8 Navier-Stokes and the corrected polynomial surface temperature basis surrogate Generalized Aerodynamic Forces (GAFs) for the HSV panel: \( f = 140 \text{ Hz}, M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa, \\
T_w = 300K, T_r = 0.085 \).

6.9 Components of the aerothremoelastic model of the HSV panel [21].

6.10 Aerothremoelastic solution using a partitioned approach to couple aerothermal and time-averaged aeroelastic solutions for the HSV panel model [21].

6.11 Two-dimensional model of the thermal structure for the HSV panel [21].

6.12 Comparison of flight times to the onset of flutter predicted using the “CIM”, “Linear”, and analytical model used by Culler and McNamara [21] (“C–M”).

6.13 Pressure, heating, panel displacement, and thermal loads during dynamically stable fluid-thermal-structural response of the HSV panel at \( M_1 = 7.5 \).

6.14 Pressure, heating, panel displacement, and thermal loads during dynamically stable fluid-thermal-structural response of the HSV panel at \( M_1 = 10.5 \).

6.15 Pressure, heating, panel displacement, and thermal loads near the onset time of flutter for both the \( M_1 = 7.5 \) (1500 sec.) and \( M_1 = 10.5 \) (40 sec) trajectories of the HSV panel.
6.16 Nonlinear fluid-thermal-structural panel flutter response at $M_1 = 7.5$. ................................................................. 173

6.17 Nonlinear fluid-thermal-structural panel flutter response at $M_1 = 10.5$. ................................................................. 174

6.18 Fluid-thermal-structural HSV panel response for the polynomial temperature basis surrogate. ........................................ 176

7.1 Comparison of several turbulent aerodynamic heating predictions to the Benchmark (CFD) for the 2-D deformable panel subject to shock impingement: $M_\infty = 3.0$, $Alt. = 24$ km, Shock Generator Angle = 9.90°. ................................................................. 183

7.2 Euler and piston theory Generalized Aerodynamic Forces (GAFs) for an oscillating panel with a stationary shock. $f_1 = 61.01$ Hz, $f_2 = 168.2$ Hz, $f_3 = 329.7$ Hz, $M_\infty = 3.0$, $Alt. = 24$ km. ............ 186

7.3 Euler and local piston theory Generalized Aerodynamic Forces (GAFs) for an oscillating panel with a stationary shock. $f_1 = 61.01$ Hz, $f_2 = 168.2$ Hz, $f_3 = 329.7$ Hz, $M_\infty = 3.0$, $Alt. = 24$ km. ............ 187

7.4 Navier-Stokes, Euler, and piston theory Generalized Aerodynamic Forces (GAFs) for an oscillating panel with a stationary shock. $f_1 = 61.01$ Hz, $f_2 = 168.2$ Hz, $f_3 = 329.7$ Hz, $M_\infty = 3.0$, $Alt. = 24$ km. . . . . . . . 188

7.5 Navier-Stokes and local piston theory Generalized Aerodynamic Forces (GAFs) for an oscillating panel with a stationary shock. $f_1 = 61.01$ Hz, $f_2 = 168.2$ Hz, $f_3 = 329.7$ Hz, $M_\infty = 3.0$, $Alt. = 24$ km. . . . . . . . 189
7.6 Navier-Stokes and pointwise surrogate-based Generalized Aerodynamic Forces (GAFs) for an oscillating panel with a stationary shock. \( f_1 = 61.01 \text{ Hz}, f_2 = 168.2 \text{ Hz}, f_3 = 329.7 \text{ Hz}, M_\infty = 3.0, \) \( \text{Alt.} = 24 \text{ km} \).

7.7 Navier-Stokes and polynomial surface temperature surrogate-based Generalized Aerodynamic Forces (GAFs) for an oscillating panel with a stationary shock. \( f_1 = 61.01 \text{ Hz}, f_2 = 168.2 \text{ Hz}, f_3 = 329.7 \text{ Hz}, M_\infty = 3.0, \text{Alt.} = 24 \text{ km} \).

7.8 Navier-Stokes, Euler, and approximate model Generalized Aerodynamic Forces (GAFs) for an oscillating shock impinging on a vibrating 2-D panel over 5 panel cycles. Shock generator: \( \pm 2^\circ \) about a nominal 10\(^\circ\) angle at 10 Hz. Panel: \( a^*_1 = 5, f = 50 \text{ Hz}, M_\infty = 3.0, \) \( \text{Alt.} = 24 \text{ km}, T_w = 433K \).

7.9 Navier-Stokes and surrogate-based Generalized Aerodynamic Forces (GAFs) for an oscillating shock impinging on a vibrating 2-D panel over 5 panel cycles. Shock generator: \( \pm 2^\circ \) about a nominal 10\(^\circ\) angle at 10 Hz. Panel: \( a^*_1 = 5, f = 50 \text{ Hz}, M_\infty = 3.0, \) \( \text{Alt.} = 24 \text{ km}, T_w = 433K \).

7.10 Structural and thermal model configurations for the 2-D shock impingement panel.

7.11 Spatially averaged through-thickness temperature rise for the unsteady, quasi-static, steady, and surrogate models.
7.12 Generalized aerodynamic forces for the unsteady, quasi-static, steady, and surrogate models. 207

7.13 Structural response for a backpressure 2000 Pa above free stream 

\[(1.673 \times P_\infty)\] for the various models. Free stream pressure = 2970 Pa. 215

7.14 Thermal response for a backpressure 2000 Pa above free stream 

\[(1.673 \times P_\infty)\] for the various models. Free stream pressure = 2970 Pa. 216

7.15 Structural and thermal response for several backpressures for the 

“Poly” model. Free stream pressure = 2970 Pa. 218

7.16 Mean error as a function of space (\(\bar{E}^*\)) for several aerodynamic heating models compared to CFD for the 3-D flat panel analysis: 

\(M_\infty = 3.0\), \(Alt. = 24km\), \(L = .254m\), \(S = .127m\). 223

7.17 Comparison of several turbulent aerodynamic heating predictions to the Benchmark (CFD) for the 3-D flat panel without shock impingement: \(M_\infty = 3.0\), \(Alt. = 24km\), \(L = .254m\), \(S = .127m\). 224

7.18 Mean error as a function of space (\(\bar{E}^*\)) for several aerodynamic heating models compared to CFD for the 3-D deformable panel analysis: \(M_\infty = 3.0\), \(Alt. = 24km\), \(L = .254m\), \(S = .127m\). 228

7.19 Comparison of several turbulent aerodynamic heating predictions to the Benchmark (CFD) for the 3-D deformable panel without shock impingement: \(M_\infty = 3.0\), \(Alt. = 24km\), \(L = .254m\), \(S = .127m\), \(h = 0.635mm\). 229
7.20 Mean error as a function of space (E∗) for several aerodynamic heating models compared to CFD for the 3-D deformable panel subject to shock impingement: \( M_\infty = 2.0, \rho_\infty = 0.713 \text{ kg/m}^3, T_\infty = 215.5\text{ K}, L = .254m, S = .127m \).

7.21 Comparison of several turbulent aerodynamic heating predictions to the Benchmark (CFD) for the 3-D deformable panel subject to shock impingement: Shock Generator=\( 7.67^\circ, M_\infty = 2.0, \rho_\infty = 0.713 \text{ kg/m}^3, T_\infty = 215.5\text{ K}, L = .254m, S = .127m, h = 0.635mm \).

7.22 Generalized Aerodynamic Forces (GAFs) for an oscillating shock impinging on a vibrating 3-D panel over 5 panel cycles. Shock generator: \( \pm 2^\circ \) about a nominal \( 10^\circ \) angle at \( 10\text{ Hz} \). Panel: \( a_1^* = 5, f = 50\text{ Hz} \). \( M_\infty = 2.0, T_\infty = 215.5\text{ K}, \rho_\infty = 0.7127\text{ kg/m}^3, T_w = 215.5\text{ K} \).

7.23 Surrogate Generalized Aerodynamic Forces (GAFs) for an oscillating shock impinging on a vibrating 3-D panel over 5 panel cycles. Shock generator: \( \pm 2^\circ \) about a nominal \( 10^\circ \) angle at \( 10\text{ Hz} \). Panel: \( a_1^* = 5, f = 50\text{ Hz} \). \( M_\infty = 2.0, T_\infty = 215.5\text{ K}, \rho_\infty = 0.7127\text{ kg/m}^3, T_w = 215.5\text{ K} \).

10.1 Residual, lift coefficient, and drag coefficient for the generic panel for laminar flow with 3 mesh densities (20k, 80k, and 320k cells). \( M_\infty = 3.0, Alt. = 24km, T_w = 220K, a_1-a_6 = 0 \).
10.2 Residual, lift coefficient, and drag coefficient for the generic panel for turbulent flow with 3 mesh densities (20k, 80k, and 320k cells).

\[ M_\infty = 3.0, \text{Alt.} = 24 km, T_w = 220 K, a_1 = 25 mm, a_2 - a_6 = 0. \] 277

10.3 Residual, lift coefficient, and drag coefficient for the HSV panel for three mesh densities (18.6k, 74k, and 298k cells). \( M_2 = 8.4, T_2 = 335 K, P_2 = 3691 \text{Pa}, T_w = 300 K, a_1^* = 15.0, a_2^* - a_6^* = 0. \] 279

10.4 Convergence of the unsteady Navier Stokes lift coefficient and drag coefficient for the HSV panel. \( M_2 = 6.86, T_2 = 300 K, P_2 = 2967 \text{Pa}, \)

\[ T_w = 900 K, a_1^* = 5 \sin(2 \pi t 140), a_2^* - a_6^* = 0. \] 280

10.5 Convergence of the unsteady Euler lift coefficient and drag coefficient for the HSV panel. \( M_2 = 6.86, T_2 = 300 K, P_2 = 2967 \text{Pa}, \)

\[ a_1^* = 5 \sin(2 \pi t 140), a_2^* - a_6^* = 0. \] 281

10.6 Residual, lift coefficient, and drag coefficient for the 2-D shock impingement panel for three mesh densities (36.7k, 147k, and 588k cells). \( M_\infty = 3.0, \text{Alt.} = 24.0 km, T_w = 220 K. \) Shock generator:

\[ = 10^\circ. \] Panel: \( a_1^* = 20.0, a_2^* - a_6^* = 0. \] 283

10.7 Convergence of the unsteady Navier Stokes lift coefficient and drag coefficient for the 2-D shock impingement panel with the stationary shock generator at \( 10^\circ. M_\infty = 3.0, \text{Alt.} = 24 km, T_w = 440 K. \) Panel:

\[ a_1^* = 5 \sin(2 \pi t 61.01), a_2^* - a_6^* = 0. \] 284
10.8 Convergence of the unsteady Euler lift coefficient and drag coefficient for the 2-D shock impingement panel with the stationary shock generator at 10°. \( M_\infty = 3.0 \), Alt. = 24km. Panel: \( a_1^* = 5\sin(2\pi t 61.01) \), \( a_2^* - a_6^* = 0 \). . . . . . . . . . . . . . . . . . . . . . . . 285

10.9 Convergence of the unsteady Navier Stokes lift coefficient and drag coefficient for the 2-D shock impingement panel with an oscillating shock generator angle: 10 Hz ±3° about a nominal 10°. \( M_\infty = 3.0 \), Alt. = 24km, \( T_w = 440K \). Panel: \( a_1^* = 5\sin(2\pi t 50) \), \( a_2^* - a_6^* = 0 \). . . . 286

10.10 Convergence of the unsteady Euler lift coefficient and drag coefficient for the 2-D shock impingement panel with an oscillating shock generator angle: 10 Hz ±3° about a nominal 10°. \( M_\infty = 3.0 \), Alt. = 24km, \( T_w = 440K \). Panel: \( a_1^* = 5\sin(2\pi t 50) \), \( a_2^* - a_6^* = 0 \). . . . . . . . . . . . . . . . . . . . . . . . 287

10.11 Residual, lift coefficient, and drag coefficient for the 3-D shock impingement panel for three mesh densities (930k, 7.4mil, and 59.5mil cells). \( M_\infty = 3.0 \), Alt. = 24.0 km, \( T_w = 220K \). Shock generator: 10°. Panel: \( a_1^* = 20.0 \), \( a_2^* - a_6^* = 0 \). . . . . . . . . . . . . . . . . . . . . . . . 290

10.12 Convergence of the unsteady Navier Stokes lift coefficient and drag coefficient for the 3-D shock impingement panel with an oscillating shock generator angle: 10 Hz ±3° about a nominal 10°. \( M_\infty = 2.0 \), \( T_\infty = 215.5K \), \( P_\infty = 44093Pa \), \( T_w = 215.5K \). Panel: \( a_1^* = 5\sin(2\pi t 50) \), \( a_2^* - a_6^* = 0 \). . . . . . . . . . . . . . . . . . . . . . . . 291

xxxvii
10.13 Convergence of the unsteady Euler lift coefficient and drag coefficient for the 3-D shock impingement panel with an oscillating shock generator angle: 10 Hz ±3° about a nominal 10°. \( M_\infty = 2.0, T_\infty = 215.5K, P_\infty = 44093Pa, T_w = 215.5K. \) Panel: \( a_1^* = 5sin(2 \pi t 50), a_2^*-a_6^* = 0. \) ................. 292

10.14 Residual, lift coefficient, and drag coefficient for the control surface for three mesh densities (34k, 270k, and 2.2mil cells). \( M_\infty = 8.0, \) Alt. = 40.0 km, \( T_w = 1200K, a_1^* = 0, a_2^* = 0. \) ................. 294

10.15 Convergence of the unsteady Navier Stokes lift coefficient and drag coefficient for the control surface. \( M_\infty = 8.0, \) Alt. = 40.0 km, \( T_w = 1200K, a_1^* = 1.0sin(2 \pi t 200), a_2^* = 0. \) ................. 295

10.16 Convergence of the unsteady Euler lift coefficient and drag coefficient for the control surface. \( M_\infty = 8.0, \) Alt. = 40.0 km, \( a_1^* = 1.0sin(2 \pi t 200), a_2^* = 0. \) ................. 296
List of Symbols

$A = \text{amplitude}$

$a = \text{speed of sound}$

$a_i = \text{modal weight normalized by thickness}$

$B = \text{sideslip angle}$

$b_i = \text{polynomial temperature coefficients}$

$C = \text{Chapman and Rubesin constant}$

$C = \text{correlation model for kriging}$

$[C] = \text{thermal capacitance matrix}$

$c = \text{specific heat}$

$c_f = \text{skin friction}$

$C_H = \text{Stanton number}$

$C_i = \text{free stream coefficients for CIM}$

$C_p = \text{pressure coefficient}$

$c_p = \text{specific heat at constant pressure}$

$Cd = \text{drag coefficient}$

CIM = Corrected Isothermal Model for heat flux
\( Cl \) = lift coefficient

\( CFD \) = Computational Fluid Dynamics

\( D \) = bending stiff of HSV panel

\( D_n \) = coefficients for CIM

\( d \) = vector of input parameters for surrogate prediction

\( DOF \) = Degrees of Freedom

\( E \) = Young’s modulus

\( \hat{E} \) = maximum dimensional error of each test case

\( \hat{E}^{\ast} \) = maximum percent error of each test case

\( E^{\ast} \) = mean percent error of each test case

\( \bar{E}^{\ast} \) = mean error of all test cases as a function of space

\( f \) = frequency

\( F(X) \) = matrix of the polynomial curve fit to the sample points,

\[
[f(X_1), ..., f(X_n)]^T
\]

\( f(\cdot) \) = vector of the polynomial curve fit to a single set of input parameters,

\[
[f_1(\cdot), ..., f_p(\cdot)]^T
\]

\( f_i \) = basis function regression model

\( F-T-S \) = Fluid-Thermal-Structural

\( G(X) \) = Gaussian correlation matrix

\( g(d) \) = Gaussian correlation vector for ROM prediction

\( GAF \) = Generalized Aerodynamic Force
$H$ = enthalpy

$H_t$ = transverse tunnel height

$h$ = thickness

$h_{emp}$ = empirically determined heat transfer coefficient

$h_T$ = heat transfer coefficient

HSV = Hypersonic Vehicle

Int = integral based heat flux model

Iso = pointwise heat flux model obtained from isothermal solutions

$[K]$ = thermal conductivity matrix

$k$ = thermal conductivity

$L$ = length

LHS = Latin Hypercube Sampling

$M$ = Mach number

$M_T$ = thermal bending moment, HSV panel

$N$ = normal direction from the wall

$N_x$ = in-plane stress resultant, HSV panel

$P$ = pressure

$P_r$ = Prandtl number

POD = Proper Orthogonal Decomposition

Poly = polynomial temperature basis for heat flux

$Q$, $Q_{aero}$ = aerodynamic heat flux
\{Q\} = nodal heat load vector

\(Q_{rad}\) = radiation heat flux

\(Q_{CIM}\) = corrected pointwise Iso model heat flux

\(Q_{ISO}\) = pointwise heat flux obtained from isothermal solutions

\(\Delta q\) = correction to isothermal heat flux

\(R\) = gas constant for air \((287.0\,J/kg/K)\)

\(R\) = regression model for kriging

\(V\) = matrix of eigenvectors

\(r\) = recovery factor

\(Re_x\) = local Reynolds number

\(\text{ROM/S}\) = Reduced Order Models/Surrogates

\(S\) = span of 3-D panel

\(\text{SSI}\) = Shock-Shock Interaction

\(\text{STBLI}\) = Shock-Turbulent Boundary Layer Interaction

\(T\) = temperature

\(t\) = time

\(T_r\) = coefficient for empirically fit local piston theory model

\(U\) = velocity

\(u\) = chordwise velocity

\(v\) = transverse velocity

\(v_n\) = normal velocity
\( W \) = width

\( w \) = surface deformation

\( \bar{w} \) = truncated set of modal weights of \( Y(X) \)

\( X \) = model configuration parameters

\( X \) = matrix input parameters over the set of sample points for ROM construction

\( X_i \) = vector of input parameter values for the \( i^{th} \) sample point

\( x \) = chordwise direction, flow-direction

\( y \) = spanwise direction

\( Y(X) \) = full-order solutions at sample points \( X \)

\( y(d) \) = reduced-order approximation to the full-order system

\( Y_n'(0) \) = coefficients for temperature distribution in Eq. (4.14)

\( Z \) = function describing surface geometry

\( z \) = transverse direction

\( \alpha \) = angle-of-attack

\( \alpha^* \) = coefficient of thermal expansion

\( \beta \) = shockwave angle

\( \gamma \) = ratio of specific heats

\( \Delta \) = increment

\( \delta \) = boundary layer displacement thickness
\( \theta \) = inclination angle

\( \theta_k \) = correlation parameter

\( \Lambda \) = matrix of eigenvalues

\( \mu \) = viscosity

\( \xi \) = estimated modal weights for a desired point in the parameter space

\( \rho \) = density

\( \sigma^2 \) = process variance

\( \phi \) = deformation modeshape

\( \psi \) = temperature modeshape

\( \Psi \) = matrix of POD modes

\( \bar{\Psi} \) = truncated set of POD modes

**Subscripts**

\( \infty \) = free stream

\( ^* \) = evaluated at Eckert’s reference enthalpy

\( 0 \) = total condition

\( AE \) = aeroelastic

\( AT \) = aerothermald

\( AW \) = adiabatic wall

\( e \) = boundary layer edge
i, j, n = indices
loc = locally computed value
p = number of data points in a single snapshot
s = shock generator
t = wind tunnel
w = at the wall or surface
x = x-direction
y = y-direction
z = z-direction

Superscripts

* = nondimensional
\dot{} = derivative with respect to time
\prime = derivative with respect to the free stream direction
\bar{} = average value
T = matrix transpose
Chapter 1

Introduction and Objectives

1.1 Introduction

The National Aeronautics and Space Administration (NASA), the Department of Defense, and manufacturers such as Lockheed Martin and Boeing have all expressed a renewed interest in responsive and reusable hypersonic cruise vehicles [1–10]. These vehicles are necessary to meet several objectives: (1) Persistent & Responsive Precision Engagement, (2) On-demand Force Projection, Anywhere, (3) To Enable Very-High Speed Flight for Launch Vehicles, and (4) To Enable Re-entry into Planetary Atmospheres [1, 2]. Demonstration and concept vehicles developed by the NASA, Defense Advanced Research Projects Agency (DARPA), and the U.S. Air Force, include: the Force Application and Launch from Continental United States (FALCON) program [5], the X-43A and X-51A scramjet vehicles [7, 8], and the Hypersonic International Flight Research Experimentation Program (HIFiRE) [4]. These programs are focused on the development and demonstration of hyper-
sonic technologies in order to enable these prompt global reach objectives.

In order to meet these objectives, typical design of these vehicles is based on lifting body configurations with an air-breathing engine, small aerodynamic control surfaces, and an integrated airframe-propulsion system [2, 4, 7, 8, 11]. An example vehicle, the NASA X-43A Hyper-X test vehicle, is illustrated [12] in Fig. 1.1. However, an extensive set of technical challenges have obstructed the development of these vehicles[1, 2, 5, 9, 10]. Many of these challenges stem from three currently incompatible design constraints: 1) operating in an extreme environment, 2) vehicle survivability, and 3) vehicle responsiveness.

![Image of NASA X-43A Hyper-X](image)

Figure 1.1: 3 view of the NASA X-43A Hyper-X hypersonic test vehicle [12].

An air-breathing capability is desired in order to improve reliability, reduce the cost of operation by obtaining the oxidizer for combustion from the atmosphere
instead of carrying it on-board, and increasing the payload by reducing the overall quantity of fuel carried [11]. This implies that these vehicles will operate in extreme environmental conditions, due to long duration, high-speed flight in the atmosphere [2, 4, 11, 13–15]. Therefore hypersonic cruise vehicles will experience significant aerothermodynamic loading from the environment, due to the aerodynamic heat flux and aerodynamic pressure acting on the surface of the vehicle [4, 6, 7, 16].

In the past, vehicle survivability was addressed by mitigating the aerodynamic heat load through ablative thermal protection systems [6]. However, this increases vehicle weight and reduces both vehicle responsiveness and re-usability. This motivates light-weight hot structures. However, such structures are expected to be compliant while also operating in an extreme, dynamic environment. Thus, the design of such structures necessitates the consideration of coupled fluid-thermal-structural (F-T-S) interactions, shown in Fig. 1.2.

The driving forces in F-T-S interactions are the fluid loads: aerodynamic surface pressure ($P$) and aerodynamic surface heat flux ($Q$). These loads act as boundary conditions for the structural and thermal components, respectively. Coupling of the components occurs when the deformation ($w$) and surface temperature ($T_w$) are fed back into the fluid as boundary conditions. These boundary conditions then change both the pressure and heat flux. Additionally, heat flux and pressure lead to transient deformations and evolving structural properties. These transient structural properties, combined with the coupled nature of the problem, result in
Figure 1.2: Fluid-Thermal-Structural Coupling.
The need to account for coupling and the path dependence of the structural response and loads in this class of vehicle implies that simulation and ultimately life prediction may need to be carried out over the entire trajectory instead of the worst case loading points in a trajectory [17]. Due to wind tunnel limitations and the inability to scale fluid-thermal-structural responses, ground-based experimental testing is impractical to meet this need [18, 19]. This indicates that computational simulation will have the primary role in the development and analysis of hypersonic vehicles. However, performing a long time-record, numerical, multi-disciplinary analysis is an arduous task considering the high computational cost and the need to account for substantial uncertainty in hypersonic fluid, thermal, and structural problems [18, 20]. Collectively, these issues imply the need for probabilistic analysis and strictly limit the formulation of multi-disciplinary models to methods that are computationally efficient. This constraint combined with the complexity and coupled nature of the hypersonic environment makes accurate and efficient aerothermodynamic loads prediction paramount to the design, analysis, simulation, and ultimately life prediction of high-speed vehicles.

As noted in Fig. 1.2, aerothermodynamic models must be capable of including the feedback mechanisms of spatially and temporally variable surface temperatures and deformations. Additionally, vehicle scale analysis will require accounting for flow non-linearities such as shock impingement, shock-shock interactions,
and three-dimensional effects. Historically, the hypersonic aerothermodynamic loads have been approximated using either computational fluid dynamic (CFD) solutions to the Navier-Stokes equations, or basic analytical approaches. CFD relies on the fewest assumptions, and provides the potential to capture the complete flow physics. However, the associated computational expense makes trajectory scale and/or probabilistic analysis intractable for a multi-disciplinary simulation. At the other end of the modeling spectrum, analytical approaches provide efficient prediction of the aerodynamic pressure and heating, albeit at the expense of several simplifying assumptions [21, 22]. A third option, in the middle of the modeling spectrum, are CFD-based reduced-order models/surrogates (ROM/S), which seek to provide an accurate description of a system at a computational cost that is a fraction of that needed for a high-fidelity analysis. However, ROM/S present another set of challenges in terms of robustness and ease of construction. In terms of robustness, the ROM/S must be built prior to a F-T-S simulation and must be capable of handling the unknown feedback effects of spatially and temporally varying surface temperatures and deformations. In terms of ease of construction, the generation of the ROM/S must be computationally efficient compared to simply implementing full-order CFD in the F-T-S analysis.
1.2 Literature Review

In order to assess the state-of-the-art and critical needs in aerothermodynamic modeling for F-T-S analysis, literature in the following areas are reviewed: (1) fluid-thermal-structural simulations, (2) aerodynamic heating, (3) aerodynamic pressure, and (4) CFD-based ROM/S.

1.2.1 Fluid-Thermal-Structural Interactions

A thorough understanding of F-T-S interactions is crucial to the development of an efficient and accurate aerothermodynamic modeling capability. A recent review paper [17] as well as several earlier studies [23–26] provide insight into the complexity of F-T-S interactions. Due to the computational cost and complexity involved, only a limited number of studies have considered the F-T-S interactions of wings and complete vehicles [17]. In order to maintain computational tractability, recent studies have neglected either both feedback mechanisms of surface temperature and deformation in computing the aerodynamic heating [27–29] or just the feedback of deformation in computing the aerodynamic heating [30].

In order to include feedback, simpler cases are generally considered, namely panels and airfoils [21, 31–35]. Thornton and Dechaumphai [31], and Dechaumphai et al.[32] coupled quasi-static finite element flow, thermal, and structural models for panels [31] and leading edges [32]. The thermal and structural loads were updated using steady-state aerodynamic pressure and heating due to quasi-static
structural deformations, thus unsteady effects were neglected. In a related study, Loehner et al. [33] coupled steady-state CFD, computational thermal dynamics, and computational structural dynamics codes. The assembled framework was used to repeat the aerodynamically heated panel study of Thornton and Dechaumphai [31]. Culler and McNamara performed dynamic, coupled F-T-S analysis for the cylindrical bending of simply-supported panels [21] and a stiffened composite panel [35]. Feedback of both deformation and surface temperature were included in the prediction of the aerodynamic heating and pressure, where the pressure and heat loads were modeled using analytical/empirical models; i.e. third-order piston theory aerodynamics [36] and Eckert’s reference enthalpy method [37].

In these studies, it was found that the aerodynamic heat flux distribution can be altered significantly by only modest panel deformations [31, 33] and by shock-shock interference heating due to leading edge deformations [32]. Additionally, including feedback of the deformation in the aerodynamic heating creates a non-uniform aerodynamic heat flux which results in non-uniform temperature distributions, material property degradations, and increased peak temperatures and surface ply failure [21, 35].

Another important loading case in high-speed flow is shock-turbulent boundary layer interactions (STBLIs), since they amplify turbulent boundary layer loads and cause high localized heating [38, 39]. Thus, STBLIs present a significant risk to severely damage the surface panels of high speed vehicles. However, shock impingement and the associated response of inflicted panels are challenging to model.
since there is a significant amount of uncertainty in modeling of STBLIs, wind tunnel testing of F-T-S response due to shock impingement is extremely difficult in the hypersonic flow regime, the impingement location is a function of the transient thermo-structural response, and the thermal response time is on the order of minutes to hours.

Recently, Visbal [40, 41] investigated the fluid-structural response of a panel in Mach 2 flow with an impinging shock for both inviscid [40] and laminar flow [41]. The analysis was carried out by coupling a CFD solver to a finite difference solver of a von Kármán plate. These studies found that the shock impingement significantly reduced the dynamic pressure required to incite panel instabilities. The panel response was also strongly dependent on the shock impingement point. In addition, Visbal [41] investigated the effects of a prescribed time-varying panel backpressure, and found that the associated panel oscillations could reduce the shock induced flow separation on the outer surface.

In experimental wind tunnel testing [42] of static flat plates, shock impingement has been found to increase the aerodynamic heat flux by a factor of 10 at Mach numbers as low as 4. Additionally, the Air Force Research Lab (AFRL) is conducting wind-tunnel experiments on thermally and structurally compliant panels in the RC-19 supersonic wind tunnel [43, 44]. Initial results indicate that the pressure field is also observably changed for a compliant panel relative to a rigid panel.

From these studies, it is clear that complex, high-speed flow fields will lead to F-T-S interactions with feedback of variable surface deformations and non-uniform
surface temperatures. Additionally, shock-shock and shock-boundary layer interactions may be present and these effects combined with the variable surface boundary conditions are critical for the accurate prediction of the aerodynamic heating and aerodynamic pressure loads. Furthermore, it is noted that feedback of the surface profile is crucial for the aerothermodynamic loads in a coupled F-T-S analysis; and by extension that three-dimensional surfaces and resulting flowfields will have a dramatically effect on these loads.

### 1.2.2 Aerodynamic Heating

In terms of modeling the aerodynamic heating, a robust model is desired that is capable of accurately accounting for nonlinear, three-dimensional flow, which may include shock-shock interaction and shock impingement. The model must also be able to account for spatially and temporally varying surface temperatures and surface deformations, which act as important feedback mechanisms from the thermomechanical response of the system [17, 21, 35]. This is particularly true for the aerodynamic heating, since it requires a highly resolved analysis of the boundary layer [18, 45].

Specifically related to supersonic/hypersonic vehicle studies, DeBonis et al. [45] compared different Reynolds Averaged Navier-Stokes (RANS) turbulence models and different RANS codes, and concluded that the dominant factor in the accuracy of the solution was the choice of the turbulence model. In a similar study, Bertin and Cummings [18] note that there are several factors affecting the accuracy of
CFD modeling of hypersonic vehicles; including: turbulence modeling for RANS
codes, transition location, grid density, the ability to model either equilibrium or
nonequilibrium flows, and finally correctly modeling vehicle geometry. A number
of turbulence models are available for closure of the RANS equations, including
Spalart-Allmaras [46], Menter $k - \omega$ SST [47], Wilcox $k - \omega$ [48], and Baldwin-
Lomax [49]. Recently, the applicability of these turbulence models was verified in
the supersonic/hypersonic regime in several studies [50–55].

Shock impingement is also a very active area of research for CFD [53, 56–60],
where considered approaches include RANS [53, 57, 60], large eddy simulation
(LES) [56, 58], and direct numerical simulation (DNS) [56, 57, 59]. As expected,
these studies have confirmed that DNS provides the best comparison with exper-
imental results, but at an extreme computational cost; whereas LES and RANS
yield higher levels of error but are more efficient. In particular, RANS cannot ac-
count for shock unsteadiness due to interactions with a turbulent boundary layer
[38, 56, 57]. However, a partial correction has been developed to account for some
of the associated effects [53]. While these works clearly highlight the challenges
associated with accurate modeling of shock impingement, important missing ef-
facts are mechanical and thermal compliance of the surface - both of which must
be considered for F-T-S analysis, and to develop accurate and efficient models for
use in design and life prediction studies of hypersonic vehicle structures. Thus,
while a RANS code may not provide the highest fidelity relative to DNS, RANS
is implemented in this study for three reasons: 1) RANS codes are more compu-
tationally efficient; 2) they provide a reasonable model of the flow fields around ultra high speed vehicles, if care is taken with the grid and turbulence modeling [18, 45]; and 3) RANS codes are more prevalent as an industrial design tool.

Though CFD solutions to the Navier-Stokes equations are capable of accounting for a number of complex effects associated with high-speed flight, even RANS based analysis is too computationally prohibitive to implement into a fully-coupled F-T-S analysis covering long time-records [17]. This constraints narrows the options for modeling the aerodynamic heating to analytical/empirical methods and ROM/S based on CFD.

A survey of the literature reveals a number of analytical and empirical models developed over the last 60 years, while there is only one study which modeled high-speed aerodynamic heating using a ROM/S [61]. In terms of the analytical/empirical models, several studies considered the effect of a non-uniform surface temperature in incompressible flow [62–66] and compressible flow [67, 68] over flat surfaces and incompressible flow over wedge type surface profiles [69, 70]. A thorough review of many of these methods and several other incompressible and compressible methods were published in 2010 [71]. However, all of these methods are for simple two-dimensional flow, where the form of the velocity flow-field must be assumed, and thus they are incapable of handling arbitrary deformation, three-dimensional flow effects, shock-shock interaction, or shock impingement.

Another approach [21, 35, 37, 72, 73] for computing the aerodynamic heating is
to compute the inviscid flow properties at the surface for use in a viscous boundary layer analysis. Commonly used models are Eckert’s reference temperature and enthalpy methods [21, 35, 37, 72, 74–78]. Eckert’s approach combines boundary layer relations from incompressible flow theory with flow properties evaluated at a reference condition to yield a semi-empirical compressible boundary layer theory. While based on 2-D flow over a flat plate, deformation effects can be accounted for by computing required inviscid flow properties from the unsteady aerodynamic pressure using isentropic flow relations [21], and three-dimensionality can be partially accounted for through application along streamlines [79, 80]. However, these methods neglect at least three potentially important effects: 1) viscous interactions, 2) shock interactions, 3) surface temperature gradients. All of these effects have a significant impact on the aerodynamic heating [72].

More recently, a number of methods have been developed primarily for hypersonic flows with reasonable accuracy [74–78, 81]. These methods range from simple flat plate approximate methods to computational codes capable of handling three-dimensional vehicles. For trajectory scale modeling, several recent studies [30, 78, 81, 82], applied aerodynamic heating models to vehicle reference configurations. While these methods may be able to account for some of the complex flow effects, such as the potential to account for deformation, they are not readily designed to do so.
1.2.3 Aerodynamic Pressure

Similar to the aerodynamic heating, accurate and efficient modeling of aerodynamic pressure loads is paramount to the design and analysis of hypersonic vehicles. Outside of a CFD-based pressure model, the most commonly used methods are inviscid approaches [17, 21]. These methods are appealing because of the substantially reduced cost. A common method is classical piston theory, which was originally developed by Lighthill [36], who noted that at high Mach numbers the shock waves and expansion fans on an airfoil form at small angles to the undisturbed flow. This implies that stream-wise gradients are small compared to gradients perpendicular to the flow. Consequently, in a two-dimensional inviscid flow, a perpendicular column of fluid stays perpendicular as it moves over the surface of a structure. Ashley and Zartarian [83] were the first to document the utility of this approach for aeroelastic analysis in high speed flows.

Another method is local piston theory [22, 84, 85], which represents a refinement of classical piston theory. It is computed by replacing free-stream flow quantities with locally computed flow quantities from a steady-state flow analysis. The potential to utilize steady-state CFD is appealing since it provides a means to capture complex flow phenomena neglected by theoretical aerodynamics, at a reduced computational cost relative to time-accurate CFD. This method can decrease the problems of classical piston theory which occur due to significant three-dimensional flow effects, and/or high Mach numbers/surface inclinations [22].
A similar approach to local piston theory, developed by Scott and Pototzky [86], is to compute the steady and unsteady components of the pressure from viscous CFD. The unsteady component is approximated by accounting for the effect of surface wash velocity on the pressure using either a transpiration boundary condition or effective mode shape deflection. Thus, two separate steady-state CFD computations are used to model the unsteady pressure. The approach was examined on a wing operating at a reduced frequency of 0.05 for Mach numbers of 5, 10, and 15, and was found to provide a significant improvement in approximating the unsteady aerodynamic loads compared to classical piston theory.

A recent study [22] reviewed and evaluated a number of different approaches for a two-dimensional diamond shaped airfoil relative to unsteady results from a Navier-Stokes CFD solver. Included in the study are: classical piston theory [36], local piston theory [84, 85], unsteady shock-expansion theory [87], Van Dyke’s second order theory [84, 88], unsteady Newtonian-impact theory [72, 89], and several other methods. The results demonstrated various levels of accuracy, but models that included viscous effects were generally the most accurate.

Several other recent works have addressed the aerodynamics of: hypersonic vehicle control surfaces with steady aerodynamics [90], an entire hypersonic vehicle with inviscid first-order piston theory [91], and an entire vehicle with semi-empirical modeling of the viscous effects [92]. These investigations and the previous [22, 85, 86] have either been restricted to a limited set of operating conditions [85, 86, 92], to simplistic geometries (i.e., airfoils) [22], or to two-dimension pres-
sure models in three-dimensional environments [91, 92]. In terms of the latter, application of a piston theory to three-dimensional lifting surfaces is questionable since it neglects cross-flow effects [17].

1.2.4 Computational Model Reduction

The need for improved aerothermodynamic modeling as well as the computational expense of CFD-based F-T-S analysis has motivated research on computational model reduction techniques such as reduced-order and surrogate models; Refs. [93] and [94] contain excellent reviews on the subject. The goal of computational model reduction is to provide a quantitatively accurate description of the dynamics of a system at a computational cost that is substantially lower than that of a full-order model. The term “order” refers to the number of computational degrees of freedom (DOF) of the model. For the specific example of hypersonic aerothermodynamic flow analysis, the full-order model might consist of a CFD Navier-Stokes analysis that is computationally expensive. Common computational model reduction techniques include proper orthogonal decomposition (POD), Volterra series, and surrogates. POD represents a spectral method, where an orthogonal modal basis is computed from snapshots of the full-order system response to relevant inputs [93]. The Volterra series method uses the assumption that the response of any nonlinear system is exactly represented by an infinite series expansion of multi-dimensional convolution integrals of Volterra kernels. A Volterra series is constructed by computing a truncated set of kernels from the full-order system
response to a set of known inputs [93]. Surrogate based approaches identify a continuous approximate function, i.e., “surrogate function” from a discrete sampling of an unknown, nonlinear function over a bounded set of inputs [95]. Methods for constructing the surrogate function include radial basis functions, neural networks, polynomial response surfaces, and kriging [95, 96].

A common feature of methods constructed from POD, Volterra kernels, or surrogates is that the model is generated through a systematic input excitation of the fluid dynamics in order to create CFD-based training solutions. A key requirement towards a sufficient model is the identification of both relevant input parameters and a corresponding bounds on the inputs. An inherent trade-off is that model accuracy and robustness generally increase with increasing input parameters and size of the parameter space, while tractability of computing the model decreases.

Due to an ability to accurately capture strong nonlinear behavior due to shocks, expansion fans, and viscous dissipation, previous research in the reduced-order modeling of supersonic and hypersonic flows has primarily utilized spectral methods [61, 93, 94, 97, 98]. These approaches approximate the full-order system using the space spanned by a set of basis functions or vectors. Typically, the orthogonal basis is computed using POD. Lucia [97] examined POD to model aerodynamic systems with strong shocks and nonlinearity in the parameter space. Tang et al. [61] examined the accuracy of a POD based reduced-order modeling for predicting the steady-state pressure and temperature distributions on the surface of a rigid hypersonic vehicle resembling the X-43, for Mach numbers ranging from 2 to
and angles of attack ranging from 0 to 30 degrees. Alonso et al. [99] used POD to model the varying location of a shock on a transonic airfoil due to changing operating conditions. Each of these studies found that POD based ROMs are suitable for accurate representation of flows with shocks and nonlinearity in the parameter space.

Another computational model reduction approach which has shown promise in the sub-sonic regime [96, 100–103] is kriging. Kriging [100, 104–111] is a promising surrogate method useful for replacing expensive computer models (i.e., CFD) with computationally efficient approximations of nonlinear functions [105, 109, 110]. Kriging does not require a priori assumptions on the form of the full-order function that is to be approximated [95]. Simpson et al. [105] performed studies on the optimization of an aerospike nozzle for several objectives using both kriging and polynomial response surfaces. The authors found kriging to outperform the response surfaces and that kriging incurred only minimal additional computational expense.

These investigations [22, 93, 94, 97, 99, 105] in hypersonic flow and results from similar work done in lower Mach number regimes [94, 96, 100–103, 112], illustrate that both POD and surrogate based approaches are promising for accurate and efficient modeling of complex fluid dynamic phenomena. While previous work in computational model reduction of these flows has produced promising results, there is a need to further the development and application of these methods for hypersonic F-T-S analysis by considering the feedback parameters of: vari-
able surface temperature and deformation; as well as applications such as: three-dimensional flow-fields, shock-shock interactions, and shock-boundary layer interactions.

1.3 Objectives of this Dissertation

The primary goal of this dissertation is the development of highly-accurate and efficient models for predicting the aerodynamic heat flux and pressure loads on supersonic and hypersonic vehicle structures. This is accomplished by systematically investigating and developing methods from simple two-dimensional flows up through three-dimensional flows with non-uniform surface temperatures, deformations, shock-shock interactions, and shock-boundary layer interactions. Due to the complex nature of the flow-fields investigated a new two-pronged model reduction approach is developed. One prong of the approach is based on developing analytical corrections to isothermal, steady-state CFD flow solutions in order to capture complex flow effects associated with spatially-varying surface temperatures and surface pressures (e.g., surface deformation, surface vibration, shock impingements, etc.). The second prong is focused on minimizing the computational expense of computing the steady-state CFD by developing an efficient reduced order model/surrogate (ROM/S). An additional goal is to exercise the developed ROM/S in F-T-S simulations for long time record responses.

The specific objectives this dissertation are:

2. Examine coupling between surface temperature and aerodynamic heating; using analytical models, compressible boundary layer theory, and CFD.

3. Develop a high-speed CFD-based convective aerodynamic heat flux model.

4. Examine coupling between surface motion and unsteady pressure in high-speed flow, by considering the separation of steady-state and unsteady pressure components.

5. Develop a high-speed CFD-based unsteady aerodynamic pressure model.

6. Incorporate the aerodynamic heating and pressure models into a long time-record F-T-S simulations.

7. Assess the computational expense and accuracy of the models relative to both full-order CFD analysis and basic analytical/empirical approaches.

The remainder of this dissertation is organization as follows: Chapter 2 presents the computational configurations considered in this dissertation. Replacing steady-state CFD with computationally efficient surrogates is outlined in Chapter 3. Chapter 4 examines the coupling between surface temperature and the heat flux. The next chapter investigates the coupling between surface motion and unsteady pressure. Chapters 6 and 7 present F-T-S simulations of a hypersonic vehicle panel and
a panel with shock impingement, respectively. The computational expense of the various models are compared in Chapter 8. The final chapter presents the principal conclusions obtained from these studies.

### 1.4 Key Novel Contributions in this Dissertation

The principal novel contributions to the state-of-the-art contained in this dissertation are summarized below:

1. Development of robust, efficient, and accurate aerodynamic heating and unsteady aerodynamic pressure models for three-dimensional supersonic / hypersonic flow-fields; capable of accounting for variable surface temperatures, variable deformations, and shock impingements based on steady-state CFD solutions.

2. For the first time, computation of long time record CFD-based fluid-thermal-structural responses. Including a simply-supported hypersonic vehicle panel and a clamped-clamped panel undergoing oscillating shock impingement.

3. Quantification of uncertainty and efficiency of variable fidelity modeling approaches.
Chapter 2

Computational Configurations

The NASA Langley CFL3D code [113, 114] is used in this study to compute the required CFD solutions to the Euler and Navier-Stokes equations. The CFL3D code uses an implicit, finite-volume algorithm based on upwind-biased spatial differencing to solve the Euler and Reynolds-Averaged Navier-Stokes (RANS) equations.

The aerothermodynamic loads from CFL3D are computed from the Stanton number \(C_H\) and pressure coefficient \(C_p\) using [113]:

\[
Q(x, y, t) = c_p U_\infty \rho_\infty C_H(x, y, t)(T_w(x, y, t) - T_0)
\]

\[
P_w(x, y, t) = \frac{1}{2} C_p(x, y, t) \rho_\infty U_\infty^2 + P_\infty
\]

Further details of CFL3D can be found in the Appendix (Section 10.1).

Aerodynamic heating and pressure studies are carried out for several different configurations: 1) two-dimensional flow over a generic panel, 2) two-dimensional flow over a deformable hypersonic vehicle panel 3) two-dimensional flow over a deformable panel subject to shock impingement, 4) three-dimensional flow over a deformable panel subject to shock impingement, and 5) three-dimensional flow
over a deformable control surface. A description of the computational domain for each configuration is provided next, however details on steady-state mesh and unsteady time step convergence for each domain are in the Appendix (Sections 10.2 to 10.6).

### 2.1 Two-Dimensional Generic Panel

The generic panel is used for broad comparison of several different methods for aerodynamic heating prediction, and also for systematic development of an analytical correction to the heat flux to account for arbitrary, non-uniform surface temperatures. The two-dimensional panel mesh is shown in Fig. 2.1. There are a total of 322,400 cells clustered near the surface and over the panel region. There are 1001 points horizontally over the surface of the panel, and 300 points distributed 2 meters up and downstream of the panel. Additionally, 279 points are exponentially distributed vertically from the surface. Both laminar and turbulent flow are considered. Three different turbulence models are included: Menter $k-\omega$ shear stress transport (SST) [47]; Wilcox $k-\omega$ [48]; and Baldwin-Lomax [49]. For the turbulent results, transition is set 1.0 meter upstream of the panel.

The considered operating ranges for this configuration are listed in Table 2.1. The range of the surface temperature, $T_w$, is set from a lower bound of the free stream temperature up to either the total temperature or 1500 K, whichever is lower. The maximum limit of 1500 K is chosen because surface temperatures for
vehicle structures are not expected to exceed this temperature for Mach numbers of 6 and lower [21]. Deformation is included through a summation of the first six sine functions and corresponding amplitudes, according to:

\[ w(x) = \sum_{i=1}^{6} a_i \sin(i\pi x^*) \]  \hspace{1cm} (2.3)

where \( a_i \) are the modal amplitudes, and \( x^* \) is the non-dimensional spatial parameter, \( x/L \) (\( L = 0.5 \) m). The ranges for the modal amplitudes are also in Table 2.1. In order to maintain moderate deformations, the first modal amplitude is set to a maximum deflection of 5% of the length of the panel. The first modal amplitude varies inside of this range, and the subsequent modal amplitudes are reduced by 4, 8, and 16 times this value up to the fourth modal amplitude. The fifth and six modal amplitudes are set to match the fourth modal amplitude.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>$L$ (m)</td>
<td>0.5</td>
</tr>
<tr>
<td>Mach No.</td>
<td>2 – 6</td>
</tr>
<tr>
<td>$\rho_\infty$ ($g/m^3$)</td>
<td>10 – 400</td>
</tr>
<tr>
<td>$T_\infty$ (K)</td>
<td>200 – 350</td>
</tr>
<tr>
<td>$T_w$ (K)</td>
<td>$T_\infty - (T_0$ or 1500)</td>
</tr>
<tr>
<td>$a_1$ (mm)</td>
<td>± 25.0</td>
</tr>
<tr>
<td>$a_2$ (mm)</td>
<td>± 6.25</td>
</tr>
<tr>
<td>$a_3$ (mm)</td>
<td>± 3.125</td>
</tr>
<tr>
<td>$a_4$ (mm)</td>
<td>± 1.563</td>
</tr>
<tr>
<td>$a_5$ (mm)</td>
<td>± 1.563</td>
</tr>
<tr>
<td>$a_6$ (mm)</td>
<td>± 1.563</td>
</tr>
</tbody>
</table>
2.2 Two-Dimensional HSV Panel

The hypersonic vehicle (HSV) panel computational mesh included in order to perform a fluid-thermal-structural response using the previously developed thermal, structural, and time integration schemes of Culler and McNamara [21]. The relatively simple configuration of a simply-supported panel undergoing cylindrical bending is considered. The panel mesh is shown in Fig. 2.2. There are a total of 297,600 cells clustered near the surface and over the 1.5 meter long panel region. There are 1001 points horizontally over the surface of the panel, and 200 points spread between the two meters upstream and half meter downstream of the panel. Additionally, 279 points are spaced in an exponentially increasing fashion vertically from the surface. Three different turbulence models are included: Menter $k - \omega$ SST, Wilcox $k - \omega$, and Baldwin-Lomax.

As shown in Fig. 2.3, the panel is located on the surface of a two-dimensional wedge, representing the forebody surface of a HSV [21]. Note the four regions of the HSV: 1) free stream, 2) behind the oblique shock resulting from the $5^\circ$ leading edge surface of the HSV, 3) the leading edge of the panel, and 4) a point of interest along the panel. In order to reduce the computational expense, the CFD mesh only includes the surface behind the oblique shock, regions 2 to 4. The flow properties behind the oblique shock are computed using oblique shock theory [87]. The boundary conditions used in the CFD model are the flow conditions at region 2. The geometry of the HSV panel is listed in Table 2.2.
Figure 2.2: HSV panel mesh.

Figure 2.3: Panel located on an inclined surface of a wedge-shaped forebody.
Table 2.2: HSV panel geometry.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Forebody Surface Inclination, Θ (°)</td>
<td>5.0</td>
</tr>
<tr>
<td>Transition to Turbulence Upstream of Panel (m)</td>
<td>1.0</td>
</tr>
<tr>
<td>Panel Length, L (m)</td>
<td>1.5</td>
</tr>
<tr>
<td>Plate Thickness, h (mm)</td>
<td>5.0</td>
</tr>
</tbody>
</table>

The free stream Mach number and altitude considered for this model are the same as the flow conditions considered in the Culler and McNamara study [21], and are listed in Table 2.3. The initial surface temperature is set to 300K, thus this value provides the lower bound for the surface temperature parameter. The upper bound is set to 1500K which is 200K higher than the surface temperatures observed in Ref. [21]. The structural deformation is parameterized using the nondimensional modal amplitudes ($a_i^* = a_i/h$) of the six sine mode shapes, where $h$ is the thickness of the panel (5mm), according to:

$$w(x) = \sum_{i=1}^{6} a_i^* h \Phi_i(x)$$  \hspace{1cm} (2.4)

where $\Phi_i$ are the first six sine functions. Sine functions are chosen to remain consistent with the previous study [21]. The bounds considered for these six modal amplitudes are listed in Table 2.3. The first modal amplitude is set to a maximum deflection of 5% of the length of the panel ($L = 1.5$ m). The subsequent modal amplitudes are reduced by 2, 4, and 10 times this value up to the fourth modal amplitude. The fifth and six modal amplitude are set to match the fourth modal
amplitude. All of these values are chosen in order to exceed the bounds seen in the previous study [21] by at least 10%.

Table 2.3: HSV panel parameter space.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach No. ($M_1$)</td>
<td>7.5 – 10.5</td>
</tr>
<tr>
<td>Altitude (km)</td>
<td>30</td>
</tr>
<tr>
<td>$T_w$ (K)</td>
<td>300 – 1500</td>
</tr>
<tr>
<td>$a_1^*$ (Non-dim)</td>
<td>± 15.0</td>
</tr>
<tr>
<td>$a_2^*$ (Non-dim)</td>
<td>± 7.50</td>
</tr>
<tr>
<td>$a_3^*$ (Non-dim)</td>
<td>± 3.75</td>
</tr>
<tr>
<td>$a_4^*$ (Non-dim)</td>
<td>± 1.50</td>
</tr>
<tr>
<td>$a_5^*$ (Non-dim)</td>
<td>± 1.50</td>
</tr>
<tr>
<td>$a_6^*$ (Non-dim)</td>
<td>± 1.50</td>
</tr>
</tbody>
</table>
2.3 Two-Dimensional Shock Impingement Panel

This configuration is motivated by a series of shock impingement studies on thermally and structurally compliant panels recently conducted in the AFRL RC-19 wind tunnel [43, 44]. The configuration examined is shown in Fig. 2.4. Two-dimensional supersonic flow is bounded by two walls to simulate the wind tunnel. A wedge, with oscillating amplitude in time, generates an oblique shock wave that impinges on a compliant panel on the opposite wall. Panel deformations create shock waves, producing shock-shock interactions (SSI) as shown. In addition, shock-turbulent boundary layer interactions (STBLI) occur when the shock impinges on the panel. Note, the implementation of supersonic flow in this configuration is not anticipated to limit the applicability of this work to hypersonic flow, since there is no fundamental difference for shock-turbulent boundary layer interactions in supersonic or hypersonic flow.

The geometry of the shock generator (wedge) and panel are defined in Fig. 2.5 and Table 2.4. The shock generator height, $h_s$, is prescribed to vary sinusoidally in time about a mean height, $h_{s0}$, at a frequency of 10 Hz, as defined by Eq. (2.5). Note that this yields a change in the shock generator angle into the flow by $\pm 3^\circ$ about a 10$^\circ$ nominal shape. This frequency was selected based on the consideration that shock impingement on surface panels of high speed aircraft may be generated by flexible, low frequency structures such as a forebody or control surface.
Figure 2.4: Schematic of the 2-D shock impingement panel and tunnel configuration with a shock-generating wedge in supersonic flow. (*SSI*: Shock-Shock Interactions, *STBLI*: Shock-Turbulent Boundary Layer Interactions)

Figure 2.5: Schematic of the geometry of the 2-D shock impingement panel and tunnel.
Table 2.4: 2-D shock impingement panel and shock generator geometry

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$L_s$ (cm)</td>
<td>8.868</td>
</tr>
<tr>
<td>$h_{s0}$ (cm)</td>
<td>0.7818</td>
</tr>
<tr>
<td>$\delta_{s_{max}}$ (cm)</td>
<td>0.2418</td>
</tr>
<tr>
<td>$L$ (cm)</td>
<td>25.415</td>
</tr>
<tr>
<td>$\Delta_p$ (cm)</td>
<td>6.412</td>
</tr>
<tr>
<td>$H_t$ (cm)</td>
<td>13.11</td>
</tr>
</tbody>
</table>

\[
h_s(t) = h_{s0} + \delta_{s_{max}} \sin\left(2\pi 10 t\right) \tag{2.5}\]

The 2-D computational mesh is shown in Fig. 2.6. There are a total of 588,000 cells clustered near the surfaces and near the shock generator and panel regions. There are 1101 points horizontally across the surface of the panel, and a total of 1581 points in the $x$-direction. Additionally, 403 points are exponentially distributed vertically from both surfaces. Note that the flow is set to transition from laminar to turbulent flow 2.0 meters upstream of the panel. Thus, both the shock generator and the panel are subject to turbulent flow. The turbulence model chosen is the Menter $k - \omega$ SST turbulence model, because it is a well established turbulence model for high speed flow [52].

The free stream properties for this configuration are based on standard atmosphere at an altitude of 24 km. The Mach number is set to 3.0, and the lower bound of the surface temperature is set to free stream. The upper bound of the surface temperature is set to 600K which is anticipated to be higher than the adiabatic
wall temperature due to the moving shock, and is just below the total temperature for these flow conditions (616K). The structural deformation of the panel is represented using the first six structural free vibration modes and non-dimensional amplitudes ($a_i^* = a_i/h$), where $h$ is the thickness of the panel (0.711mm), according to Eq. (2.4). In Eq. (2.4) $\Phi_i(x)$ represents the clamped-clamped free vibration modeshapes of this panel. The bounds considered for the six modal amplitudes are listed in Table 2.5. Similar to the previous models, the first modal amplitude is set to a maximum deflection of just above 5% of the length of the panel ($L = 25.4$ cm). The first modal amplitude varies inside of this range, and the subsequent modal amplitudes are reduced by 2, 4, and 8 times this value up to the fourth modal amplitude. The fifth and six modal amplitude are set to match the fourth modal amplitude. Based on results from the previous studies listed in this chapter,
these ranges are expected to provide reasonable bounds for this model. The shockgenerator wedge angle, which varies from approximately $7^\circ$ to $13^\circ$, and the first six fundamental frequencies of the first six modal amplitudes are also included in Table 2.5.

Table 2.5: 2-D shock impingement panel parameter space.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$L$ (cm)</td>
<td>25.415</td>
</tr>
<tr>
<td>$h$ (cm)</td>
<td>0.07112</td>
</tr>
<tr>
<td>Mach No.</td>
<td>3.0</td>
</tr>
<tr>
<td>Altitude (km)</td>
<td>24.0</td>
</tr>
<tr>
<td>$P_\infty$ (Pa)</td>
<td>2970</td>
</tr>
<tr>
<td>$T_\infty$ (K)</td>
<td>220</td>
</tr>
<tr>
<td>$\rho_\infty$ ($kg/m^3$)</td>
<td>$4.7038 \times 10^{-2}$</td>
</tr>
<tr>
<td>$T_w$ (K)</td>
<td>220 – 600</td>
</tr>
<tr>
<td>Shock-Generator Angle ($^\circ$)</td>
<td>7 – 13</td>
</tr>
<tr>
<td>$a^*_1$ (Non-dim)</td>
<td>± 20.0</td>
</tr>
<tr>
<td>$a^*_2$ (Non-dim)</td>
<td>± 10.0</td>
</tr>
<tr>
<td>$a^*_3$ (Non-dim)</td>
<td>± 5.0</td>
</tr>
<tr>
<td>$a^*_4$ (Non-dim)</td>
<td>± 2.5</td>
</tr>
<tr>
<td>$a^*_5$ (Non-dim)</td>
<td>± 2.5</td>
</tr>
<tr>
<td>$a^*_6$ (Non-dim)</td>
<td>± 2.5</td>
</tr>
<tr>
<td>$f_1$ (Hz)</td>
<td>61.01</td>
</tr>
<tr>
<td>$f_2$ (Hz)</td>
<td>168.2</td>
</tr>
<tr>
<td>$f_3$ (Hz)</td>
<td>329.7</td>
</tr>
<tr>
<td>$f_4$ (Hz)</td>
<td>544.4</td>
</tr>
<tr>
<td>$f_5$ (Hz)</td>
<td>813.2</td>
</tr>
<tr>
<td>$f_6$ (Hz)</td>
<td>1136</td>
</tr>
</tbody>
</table>
2.4 Three-Dimensional Shock Impingement Panel

A three-dimensional representation of the previous configuration is also developed, with operating conditions based on those considered in the AFRL RC-19 wind tunnel experiments [43, 44]. The CFD mesh has a total of 7.44 million cells, with clustering near the surface, the shock generator, and the panel regions. There are a total of 501 points in the $x$-direction, 123 points in the $y$-direction, and 155 points in the $z$-direction. There are 255 and 41 points in the $x$- and $y$-directions evenly spaced over the panel. Note that the flow is set to transition from laminar to turbulent flow 1.0 meter upstream of the panel. Again, both the shock generator and the panel are subject to turbulent flow. Similar to the 2-D shock impingement configuration, the turbulence model chosen for this study is the Menter $k-\omega$ SST model. The CFD mesh is shown in Figs. 2.7, 2.8, and 2.9.

The geometry of the shock generator (wedge), panel, and tunnel is schematically depicted in Fig. 2.5, with actual values listed in Table 2.6. Note, subscript $s$ indicates shock generator geometry, subscript $t$ indicates tunnel geometry, and parameters without a subscript refer to the panel.

Two different flow conditions are considered for the 3-D shock impingement panel. The first matches the conditions for the 2-D shock impingement panel, with the Mach number set to 3.0, free stream properties based on standard atmosphere at an altitude of 24 km, and the surface temperature ranging from free stream up to 600 K. The second set of flow conditions are at Mach 2.0, and approximate one of
Figure 2.7: 3-D shock impingement panel mesh.

Figure 2.8: 3-D shock impingement panel mesh (x-z view).
Figure 2.9: 3-D shock impingement panel mesh (y-z view).

Table 2.6: 3-D shock impingement panel and shock generator geometry

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>L (cm)</td>
<td>25.4</td>
</tr>
<tr>
<td>S (cm)</td>
<td>12.7</td>
</tr>
<tr>
<td>h (mm)</td>
<td>0.635</td>
</tr>
<tr>
<td>L_s (cm)</td>
<td>12.0</td>
</tr>
<tr>
<td>W_s (cm)</td>
<td>15.5</td>
</tr>
<tr>
<td>h_{s0} (cm)</td>
<td>1.058</td>
</tr>
<tr>
<td>δ_{s_{max}} (cm)</td>
<td>0.3213</td>
</tr>
<tr>
<td>Δ_p (cm)</td>
<td>5.20</td>
</tr>
<tr>
<td>L_t (m)</td>
<td>2.25</td>
</tr>
<tr>
<td>W_t (cm)</td>
<td>20.0</td>
</tr>
<tr>
<td>H_t (cm)</td>
<td>13.11</td>
</tr>
</tbody>
</table>
the conditions used in the AFRL study in Ref. [44]. Both sets of flow conditions are listed in Table 2.7. The structural deformation of the panel is the same for both flow conditions, which is represented using the first six structural free vibration modes for a clamped plate (computed from ABAQUS) along with the non-dimensional amplitudes \( a_i^* = a_i/h \), where \( h \) is the thickness of the panel (0.635mm), according to:

\[
w(x, y) = \sum_{i=1}^{6} a_i^* h \Phi_i(x, y)
\]

where \( \Phi_i(x, y) \) are the clamped free vibration modes of the panel. The bounds considered for the six modal amplitudes, and the first six fundamental frequencies, are listed in Table 2.7. The first modal amplitude is set to a maximum deflection of just above 5% of the length of the panel \((L = 25.4 \text{ cm})\). The first modal amplitude varies inside of this range, and the subsequent modal amplitudes are reduced by 2, 4, and 8 times this value up to the fourth modal amplitude. The fifth and six modal amplitude are set to match the fourth modal amplitude. The shock-generator wedge angle, which varies from approximately \( 7^\circ \) to \( 13^\circ \), is also included in Table 2.7.
Table 2.7: 3-D shock impingement panel parameter space.

<table>
<thead>
<tr>
<th>Mach 3.0 Flow Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach No.</td>
</tr>
<tr>
<td>Altitude (km)</td>
</tr>
<tr>
<td>( T_w ) (K)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Mach 2.0 Flow Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach No.</td>
</tr>
<tr>
<td>( P_\infty ) (Pa)</td>
</tr>
<tr>
<td>( T_\infty ) (K)</td>
</tr>
<tr>
<td>( \rho_\infty ) (kg/m(^3))</td>
</tr>
<tr>
<td>( T_w ) (K)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Both Flow Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shock-Generator Angle (°)</td>
</tr>
<tr>
<td>( a_1^* ) (Non-dim)</td>
</tr>
<tr>
<td>( a_2^* ) (Non-dim)</td>
</tr>
<tr>
<td>( a_3^* ) (Non-dim)</td>
</tr>
<tr>
<td>( a_4^* ) (Non-dim)</td>
</tr>
<tr>
<td>( a_5^* ) (Non-dim)</td>
</tr>
<tr>
<td>( a_6^* ) (Non-dim)</td>
</tr>
<tr>
<td>( f_1 ) (Hz)</td>
</tr>
<tr>
<td>( f_2 ) (Hz)</td>
</tr>
<tr>
<td>( f_3 ) (Hz)</td>
</tr>
<tr>
<td>( f_4 ) (Hz)</td>
</tr>
<tr>
<td>( f_5 ) (Hz)</td>
</tr>
<tr>
<td>( f_6 ) (Hz)</td>
</tr>
</tbody>
</table>
2.5 Three-Dimensional Control Surface

The final configuration considered is a three-dimensional hypersonic vehicle control surface shown in Fig. 2.10. This control surface is chosen since its aeroelastic behavior has been examined previously in hypersonic flow [30].

![Control surface geometry](image)

Figure 2.10: Control surface geometry [30].

The fluid mesh, a vertically symmetric H–H grid, is shown in Fig. 2.11. Note that cells are clustered near the surface, leading edge, and mid-chord since these locations correspond to maximum flow gradients [30]. For steady-state computations, the grid density consists of 43 points spanwise, 135 points chordwise, and 25 points extending vertically from the surface (270,000 cells). Note there was some difficulty in the construction of surrogate models for this configuration due to the choice of turbulence model. Therefore the control surface considers only the
Baldwin-Lomax turbulence model. The more complicated turbulence models can exhibit a purely numerical flow response that mimics laminar-to-turbulent transition [115]. This behavior is highly nonlinear with changes to the grid; an inherent feature of all of these studies; since the grid is modified for the unsteady results and for the generation of a CFD surrogate from steady-state solutions. All of the other configurations considered in this dissertation are panel models, in which laminar to turbulent transition regions are specified upstream of the leading edge of the panels. The control surface model is the only model in which there is no upstream region in which to specify laminar to turbulent transition, thus leading to the choice of the Baldwin-Lomax turbulence model.

![Figure 2.11: Control surface mesh.](image)

Figure 2.11: Control surface mesh.

The flow conditions considered for the control surface model are listed in Table
2.8. The free stream Mach number range is 6 to 10, and the altitude range is 25 to 45 km. Additionally, this model includes variable angles of attack, side-slip angles, surface temperature and deformation. The selected bounds for these parameters are also shown in Table 2.8, where $T_{\infty}$ is the free stream temperature at the selected altitude. These bounds were selected based on desired operating conditions, assumed linearity of the deformation, and assumed maximum surface temperature limits [116].

<table>
<thead>
<tr>
<th>Table 2.8: Control surface parameter space.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach No.</td>
</tr>
<tr>
<td>Altitude (km)</td>
</tr>
<tr>
<td>Angle of Attack, $\alpha$ ($^\circ$)</td>
</tr>
<tr>
<td>Side-slip Angle, $B$ ($^\circ$)</td>
</tr>
<tr>
<td>$T_w$ (K)</td>
</tr>
<tr>
<td>Tip Deformation, $w_{\text{tip}}/L$ (%)</td>
</tr>
<tr>
<td>$a_1^*$ (Non-dim)</td>
</tr>
<tr>
<td>$a_2^*$ (Non-dim)</td>
</tr>
</tbody>
</table>

The structural deformation is represented using the first two structural free vibration modes, Fig. 2.12, using Eq. (2.6). The bounds of the modal amplitudes are set so that the deformation at the tip of the wing is at a maximum 10% of the length ($w_{\text{tip}}/L$ is the non-dimensional deflection of the tip of the wing). The free vibration modes are approximated from a finite element representation of the Lockheed F-104 Starfighter wing [30]. The finite element model was developed in
MSC.NASTRAN by matching the model’s total mass and first bending and torsional frequencies to the corresponding F-104 wing values. A comparison of the final model values with the F-104 wing is provided in Table 2.9.

![Mode 1, first bending, 13.41 Hz.](image1)

![Mode 2, first torsion, 37.51 Hz.](image2)

Figure 2.12: First two free vibration modes of the control surface [30].

Table 2.9: Comparison of the Lockheed F-104 Starfighter wing to the control surface model [30].

<table>
<thead>
<tr>
<th>Parameter</th>
<th>F-104</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wing Mass (Kg)</td>
<td>350.28</td>
<td>350.05</td>
</tr>
<tr>
<td>1st Bending Frequency (Hz)</td>
<td>13.40</td>
<td>13.41</td>
</tr>
<tr>
<td>1st Torsional Frequency (Hz)</td>
<td>37.60</td>
<td>37.51</td>
</tr>
</tbody>
</table>
Chapter 3

CFD Model Reduction

A significant modeling challenge associated with fluid-thermal-structural interactions is the need for a time record that encapsulates the thermal response of the structure. Since the thermal response is generally expected to be on the order of minutes to hours (i.e., changing continually during the flight of a vehicle), incorporation of CFD is intractable. Additionally, inexpensive analytical methods are generally incapable of accounting for arbitrary deformation, three-dimensional flow, and shock impingement. These disadvantages have led to the development of computational reduction techniques for CFD. Here, model reduction is performed through a two-pronged modeling approach. One prong of the approach is based on developing analytical corrections to steady-state, uniform in space surface temperature CFD flow solutions in order to capture complex flow effects associated with spatially-varying surface temperatures and surface pressures (e.g., surface deformation, surface vibration, shock impingements, etc.). The second prong is focused on minimizing the computational expense associated with computing the steady-state CFD by developing an efficient reduced order model/surrogate. This
chapter is focused on the second prong, development of an efficient reduced order model/surrogate for steady-state CFD. The analytical corrections are discussed in the following two chapters.

Computational model reduction of CFD is achieved by reducing the thousands or millions of degrees-of-freedom (DOF) due to all of the variables in each cell down to just a few parameters or global DOF. Approaches considered for reducing the DOF for nonlinear flows are proper orthogonal decomposition (POD) and kriging. The framework for these methods is described next, followed by the methods themselves, and then a comparison of the methods for computing steady-state aerothermodynamic loads.

3.1 Model Reduction Framework

A flowchart of the process used in this study to model steady-state CFD data is provided in Fig. 3.1. First, the input parameters and bounds for the model are established. Latin Hypercube Sampling (LHS) is then used to identify a diverse set of sampling points. Next \( n + K \) training snap shots of the aerothermodynamic response are computed from CFD solutions to the Navier-Stokes equations at each of the sample points; \( n \) snap shots are for model construction, and \( K \) snap shots are for evaluation. If further accuracy is desired, more sample responses are added and the process is repeated. Note that a preferred approach for selecting the sampling points would be an adaptive sampling procedure, where an initial set of
sampling points is used to construct a baseline model, and subsequent points are selected systematically so as to efficiently sample the parameter space. Such a technique is often used in the optimization community [95]. However, the challenge in the context of this work is that a globally accurate model over the parameter space is sought, since it is not known \textit{a priori} where the system will operate. Existing adaptive sampling procedures are intended for the identification of an optimum configuration; thus, they are not applicable to this study. Furthermore, an initial attempt to develop an approach based on an estimated mean squared error criterion did not produce adequate results. Thus, while an important area of need, the development of such a procedure is left for future work.

Input parameters for fluid-thermal-structural (F-T-S) analysis can be categorized as follows: 1) scalar quantities, 2) functions of time, and 3) functions of time and space. Examples of the first and second categories of input include free stream Mach number, altitude, angle-of-attack, etc. The third category includes arbitrary surface deformations and surface temperatures, which are important since they strongly influence the pressure and heat loads. Since the first and second categories of input are already in parametric form, only upper and lower bounds must be defined. However, the third category requires additional steps to parameterize. A straightforward approach is to assume these functions of space and time can be represented by a summation of spatial functions multiplied by a set of time-dependent amplitudes. For example, the structural deformation can be
Figure 3.1: Schematic of process for modeling steady-state CFD data.
conveniently approximated using free vibration modes, i.e.,

\[ w(x, y, t) = a_1(t)\Phi_{ref,1}(x, y) + \ldots + a_n(t)\Phi_{ref,n}(x, y) \] (3.1)

Thus, the structural deformation is parameterized using the free vibration modal amplitudes \((a_i)\). Note, the inclusion of structural deformation in the parameter space implies a unique CFD mesh is necessary for each sample point. This process is efficiently carried out for each CFD mesh using the Hartwich and Agrawal [117] exponential decay with trans-finite interpolation mesh deformation scheme.

A similar parameterization of the surface temperature, in terms of temperature eigenmodes obtained through a homogeneous solution to the heat equation, is generally not adequate. This is due to the fact that the thermal response of the material is a strong function of flow conditions, which itself is a strong function of structural deformation and external sources (e.g., shock generators). Thus, there is a significant likelihood for excitation of temperature eigenmodes not contained within a given subset, rendering selection of a robust temperature basis an *ad hoc* procedure. As an alternative, several approaches for coupling the surface temperature and the aerodynamic heat flux are considered, including: 1) linearization of the heat flux with respect to surface temperature, 2) integral methods for the correction of the linearized heat flux, 3) a pointwise approximation, 4) using simple polynomials as a surface temperature basis, and 5) a corrected pointwise heat flux model for spatial gradients. Each is considered in the context of a CFD surrogate for aerodynamic heating in the next chapter.
3.2 Computational Model Reduction Techniques

One objective of this study is to compare kriging and POD for the computational model reduction of steady-state CFD. Note that both of these approaches utilize a series of flow snapshots in order to assemble a set of system responses to the inputs of interest. Thus, they are interchangeable within a general reduced-order modeling framework. The goal of ROM/S based analysis is to identify a computationally efficient model using a limited number of sample points of a complex non-linear function that is expensive to compute (e.g., Navier-Stokes equations) [95]. Typical prediction times are on the order of a fraction of a second [118], whereas full-order CFD solutions require on the order of minutes to hours. Thus, the benefit of ROM/S arises when the cost of the initial sampling is less than the cost of introducing the full-order model into the analysis. In the context of F-T-S analysis, the fluid-structural loads must to be updated thousands of times a second over a trajectory that spans minutes to hours [21, 35]. Furthermore, a significant amount of uncertainty is currently associated (and will be for the foreseeable future) with CFD aerothermodynamic predictions [20]. Thus, ROM/S are needed to enable the implementation of a probabilistic analysis that spans a complete trajectory.

3.2.1 Kriging

Kriging is used in the present analysis since it provides the means for non-linear interpolation, it does not require a priori assumptions on the form of the full-order
model, and has demonstrated excellent accuracy for constructing CFD surrogates [103, 105, 118].

A fundamental assumption of kriging is that two sample points that are close together in the parameter space will have similar errors. This assumption of correlated errors is appropriate since no sources of random error or “noise” exist in deterministic computer simulations [100]. A kriging surrogate for a function of interest is characterized by local deviations, $C(d, X)$, from a global approximation, $R(d, X)$. A general form of kriging is shown in Eq. (3.2) [104],

$$y(d) = R(d, X) + C(d, X) \quad (3.2)$$

where $y(d)$ is the kriging prediction at a desired point in the parameter space. Note, $d$ is a vector of input parameters at the desired sample point, and $X$ is a matrix of input parameters from the initial training sample points. For this study, the kriging surrogate is computed using the Design and Analysis of Computer Experiments (DACE) [107] toolbox in Matlab®. Following Ref. [107], the final form of the kriging predicted response, at some desired point in the parameter space, $d$, is given by Eqs. (3.3) and (3.4):

$$y(d) = \{g(d)^T G(X)^{-1} Y(X)^T \} - \{ F(X)^T G(X)^{-1} g(d) - f(d) \}^T \hat{\beta} \quad (3.3)$$

$$\hat{\beta} = \{ F(X)^T G(X)^{-1} F(X) \}^{-1} \{ F(X)^T G(X)^{-1} Y(X)^T \} \quad (3.4)$$
where \( Y(\mathbf{X}) \) is a matrix of sample responses at sample points \( (\mathbf{X}_1, ..., \mathbf{X}_n) \):

\[
Y(\mathbf{X}) = \begin{bmatrix}
CFD_1^{(1)} & CFD_1^{(2)} & \ldots & CFD_1^{(n)} \\
CFD_2^{(1)} & CFD_2^{(2)} & \ldots & CFD_2^{(n)} \\
\vdots & \vdots & \ddots & \vdots \\
CFD_p^{(1)} & CFD_p^{(2)} & \ldots & CFD_p^{(n)}
\end{bmatrix}
\]  

(3.5)

Here, \( CFD_i^{(j)} \) is the \( i^{th} \) nodal value of the CFD output for the \( j^{th} \) sample point; where \( n \) is the number of sample points taken, and \( p \) is the total number of computational degrees-of-freedom in the CFD mesh on the structure surface. \( G(\mathbf{X}_i, \mathbf{X}_j) \) is a Gaussian correlation function [100, 104, 105, 107]:

\[
G(\mathbf{X}_i, \mathbf{X}_j) = \exp\left[ -\sum_{k=1}^{m} \theta_k |\mathbf{X}_i^k - \mathbf{X}_j^k|^2 \right]
\]  

(3.6)

where \( k \) refers to the \( k^{th} \) input parameter, and \( \theta_k \) are obtained through maximizing the likelihood function:[107]

\[
\min_{\theta} \{ \phi(\theta) \equiv |G(\mathbf{X})|^{\frac{1}{n}} \sigma^2 \} \]  

(3.7)

where,

\[
\sigma^2 = \frac{(Y(\mathbf{X}) - F(\mathbf{X})\hat{\beta})^T G(\mathbf{X})^{-1} (Y(\mathbf{X}) - F(\mathbf{X})\hat{\beta})}{n}
\]  

(3.8)

The quantity \( g(d) \) is a correlation vector between the desired input \( d \), and the sample points, \( \mathbf{X} \), using the Gaussian correlation function,

\[
g(d) = [G(d, \mathbf{X}_1), G(d, \mathbf{X}_2), ..., G(d, \mathbf{X}_n)]^T
\]  

(3.9)
Both \( f(d) \) and \( F(X) \) represent a polynomial curve fit for the desired point, \( d \), and sample points, \( X \), respectively:

\[
f(d) = [f_1(d), \ldots, f_k(d)]^T
\]

\[
F(X) = [f(X_1), \ldots, f(X_n)]^T
\]

where the curve fit may be constant, linear, or quadratic:

**Constant**, \( k = 1 \):

\[
f_1(d) = 1
\]

**Linear**, \( k = m + 1 \):

\[
f_1(d) = 1, \ f_2(d) = d(1), \ldots, f_{m+1}(d) = d(m)
\]

**Quadratic**, \( k = \frac{1}{2}(m+1)(m+2) \):

\[
f_1(d) = 1, \ f_2(d) = d(1), \ldots, f_{m+1}(d) = d(m)
\]

\[
f_{m+2}(d) = d(1)^2, \ldots, f_{2m+1}(d) = d(1)d(m)
\]

\[
f_{2m+2}(d) = d(2)^2, \ldots, f_{3m}(d) = d(2)d(m)
\]

\[
\vdots, \ldots, f_k(d) = d(m)^2
\]

In this study, the constant, linear, and quadratic polynomial regression models are evaluated on a case by case basis in order to ensure maximum accuracy.

### 3.2.2 Proper Orthogonal Decomposition

POD, also denoted as the Karhunen-Loeve decomposition, is a mathematical technique that maps a full-order model to a lower order model by identifying a small
number of DOFs that adequately reproduce the behavior of a full-order model [93, 97]. It is based on singular value decomposition and extracts an “optimal” orthogonal basis for a given matrix. These basis vectors, also referred to as modes [93, 94], represent coherent structures of the flow field. The first application of POD to fluid flow was in 1967 in which POD was used to identify dominant coherent structures in turbulent flow fields from wind tunnel data [97, 119]. POD based reduction of unsteady flow fields in fluid-structural applications was initially considered by Romanowski et al. [120], wherein a POD model was constructed for a flow field described by the linearized Euler equations.

POD is used to represent the output, \( y(d) \), as a linear combination of a basis, \( \psi(x, y) \), with coefficients, \( \xi \) [116]:

\[
y(d) = \xi_1 \psi_1 + \xi_2 \psi_2 + \cdots + \xi_r \psi_r
\] (3.13)

The sample point matrix, \( Y(X) \), in Eq. (3.5) is used to compute the POD basis. The solution proceeds through the evaluation of:

\[
\frac{1}{n} Y(X)^T Y(X) V_i = \Lambda_i V_i
\] (3.14)

which yields the eigenvectors and eigenvalues of the system, where \( V_i \) indicates the \( i^{th} \) eigenvector of \( \frac{1}{n} Y(X)^T Y(X) \), corresponding to the \( i^{th} \) largest eigenvalue [121]. Note that the eigenvalues and corresponding eigenvectors are sorted in decreasing magnitude so that the dominant POD basis modes can be readily identified. The computation of the POD basis modes, \( \psi_i \), is given by Eq. (3.15) [121].

\[
\psi_i = \frac{1}{\sqrt{n \Lambda_i}} Y(X) V_i
\] (3.15)

53
The POD basis modes, $\psi$, are stored as column vectors in the POD modal matrix, $\Psi$. The number of modes retained in the POD basis can then be truncated leading to a reduced POD modal matrix, $\bar{\Psi}$, thus reducing the number of degrees of freedom in the problem. Note that both the full POD set and truncated POD set are orthogonal [121], i.e.,

$$\Psi^T \Psi = I_{n \times n}$$

$$\bar{\Psi}^T \bar{\Psi} = I_{r \times r}$$

(3.16)

where $I_{n \times n}$ represents the identity matrix of dimension $n$ and $I_{r \times r}$ refers the the identity matrix of dimension $r$. The set of modal weights, $\bar{w}$, for the truncated system is computed from:

$$Y(X) \approx \bar{\Psi} \bar{w}$$

(3.17)

$$\bar{w} = \bar{\Psi}^T Y(X)$$

(3.18)

where the orthogonality of $\bar{\Psi}$ is utilized [121]. The final step for constructing the POD model, is to generate a function to compute the truncated set of POD modal weights, $\xi$, for a desired set of input parameters. Several methods are available for the determination of this function in terms of the predefined parameter space [95], such as parametric approaches (e.g., least squares curve fit, polynomial regression, kriging, etc.) and non-parametric approaches (e.g., radial basis functions). Kriging is selected for the present study. The estimated weights of the POD modes are given by Eq. (3.19), where the truncated POD modal weights, $\bar{w}$, replace $Y(X)$. The POD-kriging approximation, $y(d)$, is completed by combining the estimated
weights, $\xi$,
\[\xi(d)^T = \{g(d)^T G(X)^{-1} \bar{w}^T\} - \{F(X)^T G(X)^{-1} g(d) - f(d)\}^T \]
\[\{F(X)^T G(X)^{-1} F(X)\}^{-1}\{F(X)^T G(X)^{-1} \bar{w}^T\} \]

with the POD modes, $\Psi$
\[y(d) = \Psi \xi(d) \] (3.20)

### 3.3 Comparison of Methods

In this section, kriging and POD are assessed for construction of models for steady-state aerodynamic heating and pressure through comparison with full-order steady-state Navier-Stokes solutions. The model chosen to evaluate the two methods is the 3-D control surface described in Section 2.5. The full set of input parameters selected for this study are: 1) free stream Mach number, 2) angle-of-attack, 3) side-slip angle, 4) altitude, 5) surface deformation, and 6) polynomial surface temperatures described by Eq. (3.21). Thus, the models will include three trajectory parameters (Mach, side-slip angle, and altitude), one control input parameter (angle-of-attack), two structural parameters ($a_i$), and six thermal parameters ($b_i$), for a total of twelve parameters. The bounds of the parameters are listed in Table 2.8. Note, for the polynomial, the constant term is set to the range listed for $T_w$ in Table 2.8, while the higher order terms are all are arbitrarily determined \textit{a priori} as: $b_1 - b_5 = \pm 650$.

\[T_w(x, y, t) = b_0(t) + b_1(t)x + b_2(t)x^2 + b_3(t)y + b_4(t)yx + b_5(t)y^2 \] (3.21)
Models are generated using between 100 to 1500 sample points, where the sample points are selected using Latin Hypercube Sampling with the criterion to maximize the minimum distance between sample points over 500 iterations. For the POD models, the first 20 modes are retained in each case, as this number consistently had the lowest errors. Errors are computed for 500 test cases, distinct from the training set using the same bounds from Table 2.8 and the polynomial temperature profiles.

Accuracy of the models is determined using several different error metrics, including the mean ($E^*$) and maximum ($\hat{E}^*$) percent errors and the maximum dimensional error ($\hat{E}$). These errors are calculated for each test case by,

$$E^*_j(\%) = \frac{1}{p} \sum_{i=1}^{p} \left( \frac{|MODEL_j^{(i)} - CFD_j^{(i)}|}{RMS_j} \right) \times 100$$ (3.22)

$$\hat{E}^*_j(\%) = \text{Max}\left( \frac{|MODEL_j - CFD_j|}{RMS_j} \right) \times 100$$ (3.23)

$$RMS_j = \sqrt{\frac{1}{p} \sum_{i=1}^{p} (CFD_j^{(i)})^2}$$ (3.24)

$$\hat{E}_j = \text{Max}\left( |MODEL_j - CFD_j| \right)$$ (3.25)

where $p$ corresponds to the total number of nodal data points in a single test case, $i$ is the node number, and $j$ corresponds to the $j^{th}$ test case. Note, the mean and max percent errors are normalized by root mean square of the CFD output. This choice of normalization is due to a number of full-order test cases in which the heat flux either crosses zero or has small variations from maximum to minimum, both of which can lead to large unrepresentative errors when normalized. Additionally,
the maximum error is also presented in dimensional form for cases with small root mean square values. Note, in order to provide scalar quantities for the three error metrics, the average value of all of the test cases is provided for $E^*$, and the overall maximum for all of the test cases is provided for both $\widehat{E^*}$ and $\widehat{E}$. Additionally, the percent of test cases with less than 5% maximum error are compared for each model constructed.

### 3.3.1 Steady-State Aerodynamic Heating

Results for the steady-state aerodynamic heating are shown in Fig. 3.2. As expected, the accuracy of the models relative to the CFD predictions increases with increasing the number of sample points. However, beyond 900 sample points, there is only a small improvement. Note that kriging generally produces a slightly more accurate model than POD for the same number of sample points.

### 3.3.2 Steady-State Aerodynamic Pressure

The accuracy of the steady-state pressure models are assessed in a similar manner to aerodynamic heating, and are shown in Fig. 3.3. Similar trends are observed for models of the pressure compared to the aerodynamic heating. However, the pressure converges more rapidly, and to lower error values. Furthermore, there is a bigger difference in accuracy between the kriging and POD-based models, with the kriging model superior in every case. Based on these results, all of the remaining surrogate models are constructed using kriging.
(a) Mean errors ($E^*$) of the 500 test cases.

(b) Average of the maximum errors ($\hat{E}^*$) of the 500 test cases.

(c) Overall maximum error ($\hat{E}^*$) of the 500 test cases.

(d) Percent of test cases with less than 5% maximum error ($\hat{E}^*$).

Figure 3.2: Mean and maximum percent errors for kriging and POD-based steady-state heat flux models.
(a) Mean errors (\(E^*\)) of the 500 test cases.

(b) Average of the maximum errors (\(\hat{E}^*\)) of the 500 test cases.

(c) Overall maximum error (\(\hat{E}^*\)) of the 500 test cases.

(d) Percent of test cases with less than 5% maximum error (\(\hat{E}^*\)).

Figure 3.3: Mean and maximum errors for kriging and POD-based steady-state pressure models.
3.3.3 Computational Expense

The computational requirements associated with ROM/S construction and implementation are critical concerns for model reduction of CFD generated loads. In Table 3.1 the computational expense of kriging, POD, and full-order Navier-Stokes are compared for the 3-D control surface for computing one iteration of the aerodynamic heating. The computational time for CFD is approximated from the time required for one time step in an unsteady analysis, starting from a converged steady-state solution.

Table 3.1: Comparison of wall times for kriging, POD, and CFD.

<table>
<thead>
<tr>
<th>Case</th>
<th>Number of Sample Points</th>
<th>CPU Time for 1 Iteration</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kriging a</td>
<td>100</td>
<td>0.0009 sec</td>
</tr>
<tr>
<td>POD a</td>
<td>100</td>
<td>0.0008 sec</td>
</tr>
<tr>
<td>Kriging a</td>
<td>1500</td>
<td>0.0052 sec</td>
</tr>
<tr>
<td>POD a</td>
<td>1500</td>
<td>0.0019 sec</td>
</tr>
<tr>
<td>CFD b</td>
<td>-</td>
<td>150 sec</td>
</tr>
</tbody>
</table>

a 1 2.60 GHz Opteron processor core, 2.0 GB RAM.
b 24 2.60 GHz Opteron processor cores, 2.0 GB RAM each.

It is evident from this comparison that the ROM/S are several orders of magnitude more efficient than the full-order CFD analysis. Furthermore, it is clear that POD offers a slight computational advantage over kriging with an increasing num-
ber of sample points; where the kriging surrogate is over 2.5 times more expensive when using 1500 sample points. Finally, note that the computational overhead required to generate sample points and train the ROM/S is relatively small when parallel processing is used. For example, computation of 1500 sample points on 240 processor cores requires approximately 13 hours.
Chapter 4

Aerodynamic Heating

Several studies are carried out for the aerodynamic heating of high speed vehicles. First, several turbulence models and analytical models are compared in order to assess uncertainty in modeling of the aerodynamic heating. Next, several different methods for coupling surface temperature to aerodynamic heating ROM/S are developed. Then, results comparing the different strategies are presented for the two-dimensional generic panel.

4.1 Turbulence and Analytical Model Analysis

As noted in the literature review, an important consideration is the uncertainty that exists due to different turbulence models. The computational configuration chosen for this comparison is the HSV panel from Section 2.2. Note, that no experimental data for the operating conditions of this geometry was available for validation. Therefore, aerodynamic heating predictions are compared using: 1) Eckert’s reference enthalpy method, 2) Eckert’s reference temperature method, 3) RANS with Menter $k-\omega$ SST turbulence model, 4) RANS with Wilcox $k-\omega$ turbulence model, and 5) RANS with Baldwin–Lomax turbulence model. Note, the details of Eckert’s
reference enthalpy method are presented in the Appendix (Section 10.7).

The surface temperature profile assumed for this comparison is shown in Fig. 4.1. Three different panel deformations are considered. The first, shown in Fig. 4.2(a), approximates a panel deformation that is influenced by both thermal buckling and aerodynamic pressure [21]. The second, shown in Fig. 4.2(c), consists of deformation in the first sine mode and approximates a panel deformation that is primarily driven by thermal buckling [21]. The third, shown in Fig. 4.2(e), consists of deformation in the third sine mode, which corresponds to panel deformations during post flutter limit cycle oscillations [21].

![Figure 4.1: Representative temperature profile used to compare several analytical and turbulence model aerodynamic heating predictions for the HSV panel.](image)

The aerodynamic heating solutions for these deformations are shown in Figs. 4.2(b), (d), and (f), respectively. In general the spatial distribution of the aerodynamic heat flux is similarly predicted for each approach. However, there are noticeable differences at the leading edge of the panel between Eckert’s methods and the CFD predictions. This is presumably due to boundary layer displacement effects not
(a) Panel shape 1, \(a_1^* = 8.8187, a_2^* = -3.1569, a_3^* = 0.1291\).

(b) Aerodynamic heat flux for panel shape 1.

(c) Panel shape 2, \(a_1^* = 10.5\).

(d) Aerodynamic heat flux for panel shape 2.

(e) Panel shape 3, \(a_1^* = 3.5\).

(f) Aerodynamic heat flux for panel shape 3.

Figure 4.2: Comparison of aerodynamic heating predictions for several HSV panel shapes, \(M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa\).
modeled by Eckert’s methods [21]. Maximum and average differences between the different models and the RANS with Menter $k - \omega$ SST turbulence model are provided in Table 4.1. Maximum differences are as high 95% and average differences are between 10 - 45%.

CFL3D is an ideal gas code, therefore it is useful to consider the approximately 20% difference between the reference enthalpy and temperature methods in Table 4.1. The reference temperature method uses an assumption of a calorically perfect gas [21], thus this difference partially represents the offset in heat flux that could be expected if a real gas model was used in the CFD analysis. However, note that this variation is within the maximum variation found by simply using different turbulence models. Thus, while using an ideal gas assumption introduces error into the present analysis, it is expected to be within the standard bounds of uncertainty for computational aerothermodynamic predictions of RANS CFD codes.

Table 4.1: Percent difference in aerodynamic heating predictions using different modeling approaches for the HSV panel.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Ref. Temp.</td>
<td>Menter</td>
<td>Menter</td>
<td>Menter</td>
<td>Menter</td>
</tr>
<tr>
<td>Max.</td>
<td>20.0%</td>
<td>62.2%</td>
<td>95.2%</td>
<td>21.0%</td>
<td>58.7%</td>
</tr>
<tr>
<td>Avg.</td>
<td>17.3%</td>
<td>21.4%</td>
<td>20.0%</td>
<td>10.6%</td>
<td>44.5%</td>
</tr>
</tbody>
</table>
4.2 Coupling Approaches for Surface Temperature and Aerodynamic Heating

The definition of aerodynamic heat flux from Fourier’s law [72] is:

$$Q \equiv k_w \left( \frac{\partial T}{\partial N} \right)_w$$

(4.1)

where \(N\) is the outward direction normal to the wall, and positive heat flux is defined as into the wall. CFD enables relatively straightforward computation of Eq. (4.1), albeit at significant computational cost within a F-T-S simulation. In general the heat flux is a complex function of free stream conditions, space, deformation, surface temperature, and model configuration:

$$Q = f(U_\infty, \rho_\infty, T_\infty, x, y, w, T_w, X_i)$$

(4.2)

where \(x\) and \(y\) are spatial coordinates, \(w\) is the structural deformation, and \(X_i\) are model specific configuration parameters (e.g., structural profile, angle-of-attack, shock impingement, etc.). Analytical and empirical approaches often neglect many of these dependencies [67] in favor of efficiency over generality. This motivates the development of CFD ROM/S, so as to maintain efficiency while also improving model generality.

Several approaches for coupling the surface temperature and the aerodynamic heat flux are considered, including: 1) linearization of the heat flux with respect to surface temperature, 2) integral methods for the correction of the linearized heat
flux, 3) a pointwise approximation, 4) using simple polynomials as a surface temperature basis, and 5) a corrected pointwise heat flux model for spatial gradients.

4.2.1 Linearization Method

Linearized heat flux approximations implement reference configurations [27–30] of a vehicle or surface in order to compute the aerodynamic heating. First, a heat transfer coefficient is obtained and then the heat transfer analysis is conducted using the convective heat flux boundary condition:

$$ Q = h_T (T_w - T_{AW}) $$

where $T_{AW}$ is the adiabatic wall temperature and $h_T$ is the heat transfer coefficient.

Comparing Eq. (4.2) and Eq. (4.3), it is clear that the heat transfer coefficient is a complex function of many parameters. A linearized approximation is obtained by computing an approximate heat transfer coefficient from two reference flow conditions [27–30]. One flow condition is used to determine the heat flux at a nominal surface temperature (e.g., free stream conditions). The second flow condition represents adiabatic conditions in order to determine the adiabatic wall temperature. The empirically fit heat transfer coefficient is then computed as:

$$ h_{emp} = \frac{Q|_{T_w=T_\infty}}{T_\infty - T_{AW}} $$

Thus, the heat flux is approximated as:

$$ Q = h_{emp}(T_w - T_{AW}) $$
where now $Q$ varies linearly with $T_w$, since $h_{emp}$ is constant with respect to surface temperature. Variable Mach number, free stream density, free stream temperature, etc. can be accommodated by developing a ROM/S for $h_{emp}$ and $T_{AW}$. Here, two CFD solutions must be generated for each sample point; one for the surface temperature set to free stream and one for the adiabatic surface temperature. Note that this doubles the number of CFD solutions necessary relative to the number of sample points. This approximation neglects the effect of variable surface temperature on the heat transfer coefficient. Additionally, the adiabatic wall temperature is a function of deformation. Thus, the effect of surface deformation on the heat flux is neglected using this approach.

### 4.2.2 Integral Methods for Aerodynamic Heating

A correction for the dependence of the heat transfer coefficient on surface temperature can be developed using integral solutions to the incompressible boundary layer energy equation, which were originally developed in the 1950s [62–71]. Note that these solutions are based on several assumptions: 1) two dimensional flow, 2) flat surfaces, 3) viscosity, thermal conductivity, density, and specific heat are independent of temperature, and 4) the velocity field is known \textit{a priori} and is independent of temperature. Additionally, these solutions incorporate changes in surface temperature in a discontinuous stepwise fashion. With these assumptions
the resulting solution for the heat flux becomes [62, 66]:

\[ Q(x) = h_T(0, x)(T_w(0) - T_{AW}) + \int_{\xi=0}^{x} h_T(\xi, x)dT_w(\xi) \] (4.6)

where the terms inside of the integral account for non-linear dependence of the heat flux on surface temperature. In Eq. (4.6), \( x \) is the surface coordinate, \( \xi = 0 \) is point where the first step change in temperature occurs, and \( T_w(0) \) is the temperature prior to the first step. A number of analytical solutions for the heat transfer coefficient, \( h_T \), have been developed for laminar incompressible flow [66]. For example, Rubesin [62] and Eckert [64] obtained for flow over a flat plate:

\[ h_T(\xi, x) = \frac{A}{x} k Pr^{1/3} Re^{1/2} x \left[ 1 - \left( \frac{\xi}{x} \right)^{3/4} \right]^{-1/3} \] (4.7)

where Rubesin [62] set the coefficient \( A \) to 0.304, while Eckert [64] set \( A \) to 0.330. For turbulent flow over a flat plate, approximations and empirical fitting [65] led to several other solutions for \( h_T \) [66]. For example, Rubesin obtained [66],

\[ h_T(\xi, x) = \frac{0.0288}{x} k Pr^{1/3} Re^{0.8} x \left[ 1 - \left( \frac{\xi}{x} \right)^{39/40} \right]^{-7/39} \] (4.8)

Several other studies have extended this formulation to flows with wedge type surface pressure gradients [69, 70] for both laminar and turbulent incompressible flow. Additionally, Ref. [66] contains a table with 12 total approximations for \( h_T \) due to the empirically fit nature of this method.

In this study, a generalized formulation is developed based on Eqs. (4.7) and (4.8). Setting \( \xi = 0 \) in these equations, eliminates the dependence of the heat transfer coefficient on surface temperature. Therefore, this portion of the analytical heat
transfer coefficient can be replaced by the empirically determined ROM/S heat
transfer coefficient, $h_{emp}$, from Eq. (4.4). Additionally, the adiabatic wall tempera-
ture ($T_{AW}$) can also be computed from a ROM/S. Thus, for laminar flow:

$$Q(x) = h_{emp}(x)(T_w(0) - T_{AW}) + \int_{\xi=0}^{x} h_{emp}(x) \left[ 1 - \left( \frac{\xi}{x} \right)^{3/4} \right]^{-1/3} dT_w(\xi) \quad (4.9)$$

and for turbulent flow:

$$Q(x) = h_{emp}(x)(T_w(0) - T_{AW}) + \int_{\xi=0}^{x} h_{emp}(x) \left[ 1 - \left( \frac{\xi}{x} \right)^{39/40} \right]^{-7/39} dT_w(\xi) \quad (4.10)$$

Note, this formulation neglects the effects of deformation on the aerodynamic heat flux.

### 4.2.3 Pointwise Approximation

An alternative to the previous two methods is a pointwise approximation for the
effect of surface temperature on aerodynamic heating. In this approach, the surface
temperature is simply incorporated as a parameter during ROM/S construction
by computing training data at different uniform in space (i.e., isothermal) temper-
atures. The ROM/S model, $Q_{ISO}$, is then a function of the surface temperature at
pointwise locations on the surface,

$$Q(x) = f(x, T_w(x)) \quad (4.11)$$

An added advantage of this approach is that it readily incorporates deformation
since the CFD training data is obtained for any general combination of input pa-
rameters. A drawback is that it does not account for surface temperature gradients
(i.e., upstream temperatures will not affect the downstream heat flux).
4.2.4 Parameterization of Surface Temperature Using Spatial Basis Functions

Spatial basis functions can be used for parameterization of the surface temperature. These basis functions take the form,

\[ T_w(x, y, t) = b_0(t) + \sum_{i=1}^{n} b_i(t)\Psi_i(x, y) \] (4.12)

where \( b_i \) are the time-varying parametric inputs for the spatial basis functions, \( \Psi_i \), in order to account for spatially varying surface temperatures. Possible basis functions include: 1) eigenmodes obtained through a homogeneous solution to the heat equation, 2) POD modes from a previously computed the thermal response, or 3) simple polynomials. In general, eigenmodes are not adequate, due to the fact that the thermal response of the material is a strong function of flow conditions, which itself is a strong function of structural deformation and external sources (e.g., shock generators). Thus, there is a significant likelihood for excitation of temperature eigenmodes not contained within a given subset, rendering selection of a robust temperature basis difficult. POD modes are more specific to the actual thermal response. However, the construction of the modal basis necessitates an aerodynamic heating model, where the relevance of the thermal basis is inherently tied to the accuracy of the heating model. This leads to either the use of a potentially inadequate flow analysis to generate the thermal POD basis, or an expensive iterative process to construct thermal and fluid ROM/S. In either case, there is no guarantee
that the coupling between the aerodynamic heating and the surface temperature is captured correctly. The use of simple polynomials is a more efficient and general approach. However, this strategy may not be appropriate for complex surface temperatures (e.g., due to shock impingements). However, for comparison with the other coupling approaches, a simple polynomial basis is considered here:

\[ T_w(x, t) = b_0(t) + \sum_{i=1}^{6} b_i(t)x^{*i} \]  

(4.13)

where \( x^{*} \) is the nondimensional value \( x/L \), and where the coefficients \( (b_i) \) are computed using a least squares fit to a surface temperature distribution computed in a heat transfer analysis. An advantage of this approach is that it incorporates spatial gradients into the heat flux prediction. However, as noted, there are several limitations associated with the need to make \textit{a priori} assumptions on the form of the surface temperature.

\section*{4.2.5 Corrected Pointwise Approach}

In order to limit the number of \textit{a priori} assumptions required to construct a CFD ROM/S, an alternative approach is developed here based on an analytical solution for aerodynamic heating from a compressible laminar boundary layer. The goal is to correct the ROM/S developed with a pointwise dependence on surface temperature for surface temperature gradient effects.
General Form of the Correction

In deriving this model, first consider the analytical compressible laminar boundary layer aerodynamic heating solution developed by Chapman and Rubesin [67]:

\[
Q(x) = \frac{k_\infty T_\infty}{2} C_w \sqrt{\frac{\rho_\infty U_\infty}{\mu_\infty x C}} \sum_{n=0}^{\infty} b_n x^n Y_n'(0)
\]  

obtained by assuming laminar flow over a flat surface, at constant \( c_p \) and \( Pr \). The Chapman-Rubesin constant \( C \), represents an assumed linear relationship between the viscosity ratio and the temperature ratio, such that:

\[
\frac{\mu}{\mu_\infty} = C \frac{T}{T_\infty}
\]  

where \( C \), is computed using an average surface temperature, \( T_w \) [67],

\[
C = \sqrt{\frac{T_w}{T_\infty}} \left( \frac{T_\infty + S}{T_w + S} \right)
\]  

This simplifies the solution of the boundary layer equations by assuming the velocity profile of the boundary layer matches that of the classical incompressible Blasius solution [67]. Another parameter in Eq. (4.14), \( C_w \), is related to the Chapman and Rubesin constant:

\[
C_w = \sqrt{\frac{T_w}{T_\infty}} \left( \frac{T_\infty + S}{T_w + S} \right)
\]  

The \( b_n \) coefficients in Eq. (4.14) represent coefficients of polynomials, which are used to approximate the difference between the surface temperature and the adia-
batic surface temperature [67]:

$$T_w^* - T_{AW}^* = \sum_{n=0}^{\infty} b_n x^n$$

$$T_{AW}^* = 1 + 0.845 \frac{\gamma - 1}{2} M_\infty^2$$  \hspace{1cm} (4.18)

where $T^*$ is the nondimensional temperature defined by $T/T_\infty$. Note that the computed values of the coefficients, $Y_n'(0)$, are listed in Table 4.2 [67].

<table>
<thead>
<tr>
<th>$n$</th>
<th>$Y_n'(0)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>-0.5915</td>
</tr>
<tr>
<td>1</td>
<td>-0.9775</td>
</tr>
<tr>
<td>2</td>
<td>-1.1949</td>
</tr>
<tr>
<td>3</td>
<td>-1.3680</td>
</tr>
<tr>
<td>4</td>
<td>-1.4886</td>
</tr>
<tr>
<td>5</td>
<td>-1.5975</td>
</tr>
<tr>
<td>10</td>
<td>-2.0121</td>
</tr>
</tbody>
</table>

Note, the full derivation of the Chapman and Rubesin compressible laminar aerodynamic heating model is included in the Appendix (Section 10.8). There are however several drawbacks to this approach. First, it is developed for laminar flow. Second, it assumes the difference between surface temperature and adiabatic temperature in the form of polynomial distributions in Eq. (4.18). Third, the use of velocity profiles from the incompressible Blasius solution is questionable. Despite these limitations, manipulation of the Chapman and Rubesin solution enables the
derivation of a correction for spatial gradients in heat flux predictions. First, note
that when setting each of the $Y_n'(0)$ coefficients in Eq. (4.14) equal to $Y_0'(0)$, the
Chapman and Rubesin solution for heat flux, Eq. (4.14), reduces to:

$$Q(x) \approx \frac{k_\infty T_\infty}{2} C_w \sqrt{\frac{\rho_\infty U_\infty}{\mu_\infty xC}} \sum_{n=0}^{\infty} b_n x^n$$ (4.19)

Substituting Eq. (4.18) into Eq. (4.19) and rearranging:

$$Q(x) \approx Q_{ISO} = \frac{1}{2} \sqrt{\frac{\rho_\infty U_\infty}{\mu_\infty xC}} k_\infty C_w Y_0'(0) \left(T_w - T_{AW}\right)$$ (4.20)

which is the Chapman and Rubesin analytical solution for heat flux of laminar
compressible boundary layers, where the effect of spatial gradients are ignored.
Thus, it represents an analytical expression for the pointwise heat flux, $Q_{ISO}$.

Next, assume the heat flux is decomposed into two components:

$$Q = Q_{ISO} + \Delta q$$ (4.21)

where $Q$ is the actual heat flux, $Q_{ISO}$ is the heat flux dependent on pointwise sur-
face temperature changes, and $\Delta q$ represents a correction for the effect of surface
temperature gradients. The form of the correction term, $\Delta q$, is determined by sub-
tracting Eq. (4.19) from Eq. (4.14):

$$\Delta q \equiv Q - Q_{ISO} = \frac{k_\infty T_\infty}{2} C_w \sqrt{\frac{\rho_\infty U_\infty}{\mu_\infty xC}} \sum_{n=1}^{\infty} b_n x^n \left(Y_n'(0) - Y_0'(0)\right)$$ (4.22)

where the infinite series now begins with $n = 1$. Collecting terms:

$$\frac{2\Delta q}{k_\infty T_\infty C_w \sqrt{\frac{\mu_\infty xC}{\rho_\infty U_\infty}}} = \Delta q^* = \sum_{n=1}^{\infty} b_n x^n \left(Y_n'(0) - Y_0'(0)\right)$$ (4.23)
Next, taking the differential with respect to $x^*$,

$$
\frac{\partial \Delta q^*}{\partial x^*} = \sum_{n=1}^{\infty} nb_n x^n \left( Y_n'(0) - Y_0'(0) \right)
$$

(4.24)

Computing the derivatives of $T_w^* - T_{AW}^*$ from Eq. (4.18), and noting that $T_{AW}^*$ is constant with respect to $x^*$, yields:

$$
\frac{\partial T_w^*}{\partial x^*} = \sum_{n=1}^{\infty} nb_n x^{n-1}
$$

$$
\frac{\partial^2 T_w^*}{\partial x^{n-2}} = \sum_{n=2}^{\infty} n (n-1) b_n x^{n-2}
$$

(4.25)

Furthermore the infinite series in Eq. (4.26) also decays to zero for $n > 2$, due to the
Expanding the left-hand side of Eq. (4.26):

\[
\frac{\partial \Delta q^*}{\partial x^*} = \frac{2}{k_\infty T_\infty C_w} \sqrt{\mu_\infty C \left( \frac{\partial \Delta q(x)}{\partial x} - \frac{\Delta q(x)}{C_w} \frac{\partial C_w}{\partial x} \right)}
\]  (4.28)

then combining Eq. (4.26) and Eq. (4.28) yields a differential equation for \( \Delta q(x) \),

\[
\frac{2}{k_\infty C_w} \sqrt{\mu_\infty C} \left( \frac{\partial \Delta q(x)}{\partial x} - \frac{\Delta q(x)}{C_w} \frac{\partial C_w}{\partial x} \right) = \sum_{n=1}^{\infty} \frac{\partial^n T_w}{\partial x^n} x^{(n-1)} \Upsilon_n
\]  (4.29)

where if there are no temperature gradients for \( x < 0 \), the boundary condition is \( \Delta q(0) = 0 \).

During a preliminary assessment in the accuracy of the developed correction, the predicted heat flux generally exhibited 15% error compared to 100 CFD solutions for laminar flow with widely varying surface temperature profiles. This error is attributed the use of the Chapman-Rubesin constant, \( C \), which enable closed-form solution of the compressible boundary layer solutions by invoking the velocity profile in the from of the incompressible Blasius solution [67]. Considering this, and the fact that Eq. (4.29) is not applicable to turbulent flow, a generalized form is developed:

\[
\frac{1}{\rho_e C_1 T_e C_2 U_e C_3} \left( \frac{\partial \Delta q(x)}{\partial x} - D_0 \Delta q(x) \right) = \sum_{n=1}^{\infty} \frac{\partial^n T_w}{\partial x^n} x^{(n-1)} D_n
\]  (4.30)

where the ideal gas law and Sutherland’s law [72] enable all of the free stream properties in Eq. (4.29) to be replaced as unknown functions of \( \rho_e, T_e, \) and \( U_e, \) and the parameters \( C, C_w, \) and \( \Upsilon_n \) are replaced with the \( D_i \) coefficients. Note, the free stream properties have been replaced by boundary layer edge properties in order to account for the effects of deformation and shock impingement. In the absence
of deformation or shock impingement, it is assumed here that the edge properties are equivalent to the free stream properties. Additionally, an analytical determination of the $C_i$ and $D_i$ terms is not possible in general. Thus, these parameters are determined numerically using systematic CFD flow analyses described next.

**Numerical Determination of $C_i$ and $D_n$**

Four analyses are used to compute estimates for $C_i$ and $D_n$, each of which consider both laminar and turbulent flow of flat generic panel configuration described in Section 2.1. The turbulent flow cases are computed using the Menter turbulence model. For this analysis, the boundary layer edge properties are assumed to be equivalent to the free stream properties. Therefore, in order to determine the three exponents, $C_i$, the free stream density, temperature, and velocity are varied independently in three separate CFD analyses. For these analyses a single non-uniform surface temperature profile is assumed. The fourth analysis holds the free stream parameters constant, while considering several non-uniform surface temperature profiles in order to determine the $D_n$ coefficients.

For these analyses $\Delta q(x)$ is determined by the difference between the actual CFD heat flux solutions ($Q$) and the pointwise CFD ROM/S for the heat flux ($Q_{ISO}$),

$$\Delta q(x) = Q(x) - Q_{ISO}(x)$$

(4.31)

CFD pointwise kriging surrogates, $Q_{ISO}$, are computed for the first three analyses by generating 50 laminar and 50 turbulent CFD training solutions for each anal-
ysis, simultaneously varying both the corresponding free stream parameter and the isothermal surface temperature. For the fourth analysis, only 10 CFD training solutions are required for the single parameter considered, isothermal surface temperatures.

First, the dependence of the correction on the boundary layer edge density, $C_1$, is determined by computing 10 laminar and turbulent CFD test cases using the assumed temperature profile shown in Fig. 4.3. The Mach number and free stream temperature are set to 3.0 and 216 K, respectively, while the free stream density is varied evenly over the 10 cases from 10.0 to 375.0 g/m$^3$. The results are shown in Fig. 4.4 for both laminar and turbulent flow, where it was determined using the method of least squares that the profiles for $\Delta q(x)$ approximately collapse when divided by $\rho e^{C_1}; C_1 = 0.504$ for laminar flow and 0.687 for turbulent flow. Note that for turbulent flow, there remains approximately a 1% spread in the profiles. This is likely due to the fact that the laminar to turbulent transition point was observed to vary between 0.75 m to 1.0 m upstream of the surface for the 10 test cases.

![Figure 4.3: Temperature profile.](image)
Figure 4.4: Correction ($\Delta q$) dependence on boundary layer edge density ($C_1$) for laminar and turbulent flow. $\rho_{\infty} = 10.0 - 375.0 \text{ g/m}^3$, $M_{\infty} = 3.0$, $T_{\infty} = 216K$. 

(a) Laminar heat flux for various densities.

(b) Turbulent heat flux for various densities.

(c) Laminar, actual - pointwise heat flux ($\Delta q$).

(d) Turbulent, actual - pointwise heat flux ($\Delta q$).

(e) Laminar collapsed $\Delta q$.

(f) Turbulent collapsed $\Delta q$. 

80
Next, the dependence of the correction on the boundary layer edge temperature, $C_2$, is considered by computing 10 laminar and turbulent CFD test cases again using the assumed temperature profile shown in Fig. 4.3. The velocity and free stream density are set to 1,180 m/s and 46.94 g/m³, respectively, while the free stream temperature is varied evenly over the 10 cases from 210 to 340 K. The results are shown in Fig. 4.5 for both laminar and turbulent flow. Using the same process as before, it was determined that the profiles for $\Delta q(x)$ approximately collapse when divided by $T_e C_2$; $C_2 = 0.474$ for laminar flow and 0.688 for turbulent flow. Unlike the turbulent flow density analysis, there is only a 0.2% spread in the collapsed profiles, due to a consistent laminar to turbulent transition point in these test cases.

The third analysis considers the dependence of the correction on the boundary layer edge velocity, $C_3$, by computing 14 laminar and turbulent CFD test cases using the assumed temperature profile shown in Fig. 4.3. Here, the free stream density and temperature are held constant at 46.94 g/m³ and 216 K, respectively. The free stream Mach number is incremented at 0.25 intervals from 2.5 to 5.75. The results are shown in Fig. 4.6 for both laminar and turbulent flow. Again using the same process, it was determined that the profiles for $\Delta q(x)$ collapse when divided by $U_e C_3$; $C_3 = 0.419$ for laminar flow and 0.106 for turbulent flow.

The last set of coefficients required from Eq. (4.30) are $D_0$ and $D_n$, which correspond with the effect of surface temperature derivatives on the correction. For this study the free stream density, temperature, and Mach number are all held constant.
Figure 4.5: Correction ($\Delta q$) dependence on boundary layer edge temperature ($C_2$) for laminar and turbulent flow. $T_\infty = 210 - 340 \, K$, $U_\infty = 1,180 \, m/s$, $\rho_\infty = 46.949 \, g/m^3$. 

82
Figure 4.6: Correction ($\Delta q$) dependence on boundary layer edge velocity ($C_3$) for laminar and turbulent flow. $a_\infty = 295.0 \text{ m/s}$, $M_\infty = 2.5 - 5.75$, $\rho_\infty = 46.94 \text{ g/m}^3$, $T_\infty = 216 \text{ K}$. 

(a) Laminar heat flux for various Mach numbers.
(b) Turbulent heat flux for various Mach numbers.
(c) Laminar, actual - pointwise heat flux ($\Delta q$).
(d) Turbulent, actual - pointwise heat flux ($\Delta q$).
(e) Laminar collapsed $\Delta q$.
(f) Turbulent collapsed $\Delta q$. 

$a_\infty = 295.0 \text{ m/s}$, $M_\infty = 2.5 - 5.75$, $\rho_\infty = 46.94 \text{ g/m}^3$, $T_\infty = 216 \text{ K}$. 

83
at: 46.94 \text{g/m}^3, 216 \text{ K}, \text{ and } M_\infty = 3.0, \text{ respectively. The surface temperature profile is varied for 15 CFD test cases for both laminar and turbulent flow. In order to identify the } D_i \text{ terms, the left hand side of Eq. (4.30) is computed for each test case. As discussed, the } D_n \text{ coefficients rapidly converge to zero. Thus, it is assumed that the first four (} D_1, D_2, D_3, \text{ and } D_4) \text{ are adequate to compute heat flux correction. Solving for these coefficients requires numerical computation of the first through fourth order spatial derivatives of the 15 different surface temperature profiles. The coefficients, } D_0-D_4, \text{ are then computed in a least squares sense over the 15 test cases.}

One of the limitations to the numerical analysis is that it is carried out using a RANS based code, which relies on turbulence models. Thus, it is important to characterize the sensitivity of the computed coefficients to turbulence modeling. This is carried out in part by considering two additional models, namely the Wilcox \( k-\omega \) two-equation model and the Baldwin-Lomax zero-equation model. The resulting values for the coefficients from this analysis is shown in Table 4.3. Note that there are relatively small differences in the coefficients for both the Menter and Wilcox turbulence models, whereas the coefficients for the Baldwin-Lomax model is most similar to those computed for laminar flow. This is likely due to the fact that there are relatively small differences in the heat flux predicted using Menter and Wilcox turbulence models, while the Baldwin-Lomax model prediction is closer in magnitude to a laminar flow prediction. These trends indicate that the magnitude of the coefficients is directly related to the magnitude of the heat flux.
Table 4.3: Approximated exponents and coefficients for Eq. (4.30) for laminar and turbulent flow.

<table>
<thead>
<tr>
<th>Exponents</th>
<th>Laminar</th>
<th>Turbulent</th>
<th>Turbulent</th>
<th>Turbulent</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Baldwin-Lomax</td>
<td>Wilcox</td>
<td>Menter</td>
</tr>
<tr>
<td>$C_1$</td>
<td>0.5040</td>
<td>0.5203</td>
<td>0.6773</td>
<td>0.6868</td>
</tr>
<tr>
<td>$C_2$</td>
<td>0.4744</td>
<td>0.4930</td>
<td>0.6416</td>
<td>0.6880</td>
</tr>
<tr>
<td>$C_3$</td>
<td>0.4190</td>
<td>0.4065</td>
<td>0.1322</td>
<td>0.1060</td>
</tr>
<tr>
<td>$D_0$</td>
<td>5.0368</td>
<td>5.3086</td>
<td>4.5135</td>
<td>4.0316</td>
</tr>
<tr>
<td>$D_1$</td>
<td>-0.5950</td>
<td>-0.6660</td>
<td>-4.1999</td>
<td>-4.0268</td>
</tr>
<tr>
<td>$D_2$</td>
<td>-0.0384</td>
<td>-0.0431</td>
<td>-0.2754</td>
<td>-0.2583</td>
</tr>
<tr>
<td>$D_3$</td>
<td>0.0007</td>
<td>0.0007</td>
<td>0.0048</td>
<td>0.0046</td>
</tr>
<tr>
<td>$D_4$</td>
<td>-0.0003</td>
<td>-0.0004</td>
<td>-0.0022</td>
<td>-0.0020</td>
</tr>
</tbody>
</table>
Corrected Pointwise Isothermal Model (CIM)

The final form of the corrected pointwise isothermal heat flux \( Q_{CIM} \) is,

\[
Q_{CIM}(x) = Q_{ISO}(x) + \Delta q(x)
\]  

(4.32)

where \( Q_{ISO} \) is computed using a CFD surrogate pointwise dependent on temperature, and \( \Delta q(x) \) is computed from the first order differential equation in Eq. (4.30) with the coefficients and exponents listed in Table 4.3. Note, when including only the \( D_n \) coefficients up to 4, Eq. (4.30) reduces to:

\[
\frac{1}{\rho_\infty C_1 T_\infty C_2 U_\infty C_3} \left( \frac{\partial \Delta q(x)}{\partial x} - D_0 \Delta q(x) \right) = \sum_{n=1}^{4} \frac{\partial^n T_w}{\partial x^n} \frac{x^{(n-1)}}{(n-1)!} D_n
\]  

(4.33)

4.3 Comparison of the Surface Temperature Coupling Strategies

Several studies are conducted for the rapid prediction of aerodynamic heating in high speed flow. Kriging surrogates using the different surface temperature coupling strategies are compared to CFD solutions of laminar and RANS equations, considering broad variations in surface temperature, deformation, free stream density, free stream temperature, free stream velocity for the generic panel configuration. First, flat surfaces are used to compare the coupling methods for both laminar and turbulent flow. Then, surface deformation is included in the comparison. In order to determine the error introduced by a surrogate representation of the full-
order model, verification of the surrogates is performed by a comparison with full-order CFD test cases inside the parameter space used to construct the surrogate. Next, in order to determine the error involved in the coupling strategies, the aerodynamic heating surrogates are compared to 100 laminar and turbulent CFD test cases with non-uniform surface temperatures, and variable free stream densities, temperatures, and velocities.

4.3.1 Construction and Verification of CFD Surrogates for Flat Surfaces

Laminar and turbulent surrogate models are constructed using the five surface temperature coupling strategies listed in Table 4.4. The models are named in accordance with the treatment of surface temperature coupling in the aerodynamic heating model. Accuracy of the models is determined using Eqs. (3.22) to (3.25).

Linearized Model

The linearized heat flux with respect to surface temperature approximation (“Linear”) is computed using a surrogate constructed from 250 sample points, requiring a total of 500 CFD training solutions. The surrogate model is used to approximate both the heat transfer coefficient, Eq. (4.4), and the adiabatic wall temperature in order to compute the aerodynamic heat flux in Eq. (4.5). The sample points are chosen through Latin Hypercube Sampling (LHS) of the input parameters: Mach number, free stream density, and free stream temperature. The bounds for these parame-
Table 4.4: Aerodynamic heating models and descriptions.

<table>
<thead>
<tr>
<th>Model</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>Linear heat flux with respect to surface temperature; computed from heat transfer coefficient; neglects deformation.</td>
</tr>
<tr>
<td>Int</td>
<td>Corrected “Linear” model with integration methods to account for variable surface temperatures; neglects deformation.</td>
</tr>
<tr>
<td>Iso</td>
<td>Pointwise heat flux modeled with isothermal surface temperature training cases.</td>
</tr>
<tr>
<td>Poly</td>
<td>Heat flux modeled with polynomial surface temperature profile training cases.</td>
</tr>
<tr>
<td>CIM</td>
<td>Corrected pointwise “Iso” model for surface temperature gradients.</td>
</tr>
</tbody>
</table>

Parameters are listed in Table 2.1. The accuracy of the surrogate model over this parameter space is verified by generating an additional 25 evaluation sample points (i.e., 50 CFD solutions) over the same parameter space (but at different points), and obtaining the 25 full-order adiabatic surface temperatures and the 25 full-order heat transfer coefficients using Eq. (4.4).

The resulting errors for laminar flow between the surrogate and full-order prediction of the heat transfer coefficient are a mean of 3.48% and a max of 5.44%. The maximum dimensional error of the heat transfer coefficient is 1.4 $W/m^2K$, corresponding to a test case with a peak heat transfer coefficient of -37.3 $W/m^2K$. The errors for the prediction of the adiabatic surface temperature are a mean of 2.73% and a maximum of 4.96%. The maximum dimensional error of the adiabatic wall
temperature is 97.2 K, corresponding to a test case with a maximum surface temperature of 1971 K. The errors for turbulent flow are similar to that of laminar flow. The errors of the heat transfer coefficient are a mean of 2.59% and a max of 2.96%, while the errors for the adiabatic surface temperature are a mean of 3.1% and a max of 4.68%.

Integral Model

The integration method to correct the linearized model (“Int”) is also considered. This model uses the both heat transfer coefficient and adiabatic wall temperature surrogates from the “Linear” model in Eqs. (4.9) and (4.10). Therefore, no additional verification of the surrogates relative to full-order CFD is necessary for this method.

Pointwise Model

Next, a surrogate for the pointwise approximation, $Q_{ISO}$, is constructed, and is denoted as the “Iso” model. As a demonstration of the difference between the “Linear” model and the “Iso” model, a series of sample points are generated for both laminar and turbulent flow. Mach numbers of 3, 4, and 5 are considered, along with the surface temperatures set to free stream, adiabatic temperature for each Mach number, and 8 points evenly distributed between these two surface temperatures. The “Linear” model is created only from surface temperatures of free stream and adiabatic temperature at each Mach number, while the “Iso” model
uses all 10 of the sample points at each Mach number. The results at the mid-chord of the panel for laminar flow are shown in Fig. 4.7 and for turbulent flow in Fig. 4.8. For both laminar and turbulent flow, mean and max differences increase with Mach number; in part due to the increasing adiabatic surface temperatures which means the “Linear” model is linearizing the heat flux over a larger region of surface temperatures.

Figure 4.7: Mid-chord aerodynamic heating for laminar flow for the generic panel, for the “Linear” heat flux model and the pointwise model (“Iso”). Alt. = 24 km.

For the generic panel without deformation, the “Iso” surrogate is constructed
Figure 4.8: Mid-chord aerodynamic heating for turbulent flow for the generic panel, for the “Linear” heat flux model and the pointwise model (“Iso”). Alt. = 24 km.
from 500 CFD training solutions. Input parameters for the model are: Mach number, free stream density, free stream temperature, and isothermal surface temperature. Verification of the model is carried out using 25 full-order isothermal CFD solutions from the same parameter space, where comparisons are made for both laminar and turbulent flow. The resulting errors for the laminar flow test cases are a mean of 14.3% and a max of 125%. The maximum dimensional error is 0.242 W/cm², corresponding to a test case with a peak heat flux of 3.22 W/cm². The errors for turbulent flow are a mean of 38.6% and max of 739%. The maximum dimensional error is 0.657 W/cm² for a case with a peak heat flux of 41.8 W/cm². Note, due to nearly constant, small magnitude values of the heat flux from several of the full-order isothermal CFD solutions, the normalized errors for both the laminar and turbulent surrogates are unrepresentative of the quality of the model. A comparison of the maximum dimensional error to the peak heat flux for that case results in a maximum differences of 7.5% for laminar flow and 1.6% for turbulent flow.

**Parameterization of the Surface Temperature using a Polynomial Basis**

Additionally, a heat flux surrogate is computed by parameterizing the surface temperature in terms of a polynomial basis. This method is denoted as the “Poly” model. The input parameters for this model are: Mach number, free stream density, and free stream temperature, and the coefficients, $b_i$, of the polynomial functions used to parametrize the surface temperature in Eq. (4.13). Prior to generating
training data, the bounds on the $b_i$ coefficients must be set. The bound on $b_0$ is set in accordance with the original bounds of the surface temperature listed in Table 2.1. The bounds on $b_1 - b_6$ are arbitrarily set at ±1000.

Table 4.5: Bounds of the $b_i$ coefficients.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$b_0$ (K)</td>
<td>200 – 1500</td>
</tr>
<tr>
<td>$b_1$ (K)</td>
<td>± 1000</td>
</tr>
<tr>
<td>$b_2$ (K)</td>
<td>± 1000</td>
</tr>
<tr>
<td>$b_3$ (K)</td>
<td>± 1000</td>
</tr>
<tr>
<td>$b_4$ (K)</td>
<td>± 1000</td>
</tr>
<tr>
<td>$b_5$ (K)</td>
<td>± 1000</td>
</tr>
<tr>
<td>$b_6$ (K)</td>
<td>± 1000</td>
</tr>
</tbody>
</table>

Similar to the other surrogates, this model is constructed from 500 CFD training solutions. Evaluation of the surrogate representation of the full-order model is conducted relative to 25 full-order CFD solutions over the same parameter space. The resulting laminar flow errors are a mean of 29.0% and a max of 620%. The maximum dimensional error is 6.64 $W/cm^2$ for a case with a peak heat flux of 6.56 $W/cm^2$. The turbulent results are a mean error of 8.83% and a max error of 492%. The maximum dimensional error is 39.0 $W/cm^2$ for a case with a peak heat flux of 20.3 $W/cm^2$. Note that these errors are fairly high due to the fact that there are two few training solutions for the parameter space. Note that the parameter space grows considerably as parameters are added; in this case the coefficients in the polynomial basis. This highlights an important disadvantage of using spatial basis functions
for incorporating surface temperature, since computation of the surrogate model is relatively expensive.

**Corrected Pointwise Isothermal Model**

The corrected pointwise “Iso” model (“CIM”) is also examined for both the laminar and Menter coefficients from Table 4.3. Note from Eq. (4.32) that this model corrects a pointwise model. Thus, “Iso” surrogate is used for the CIM, and no further evaluation of surrogate accuracy is necessary for this method.

4.3.2 **Comparison of Aerodynamic Heating Models for Laminar Flow over a Flat Surface**

In order to characterize the different modeling approaches for surface temperature feedback, 100 CFD evaluation cases with variable free stream properties and spatially varying surface temperatures are computed for laminar flow over the generic panel. None of these cases overlap with those used to construct the various surrogate models. A diverse set of surface temperature profiles are specified by:

\[ T_w(x) = T_0 + \sum_{i=1}^{8} T_i \Psi_i(x) \]  

(4.34)

where \( T_0 \) is an isothermal surface temperature, and \( \Psi_i(x) \) are 8 shape functions normalized between 0 and 1. These functions include: second through fifth order polynomials, two Gaussian pulses, a sine function, and a cosine function. The bounds of \( T_0 \) are the same as the bounds on \( T_w \) from Table 2.1, and the bounds of
$T_1-T_8$ are $\pm 100$. The test cases are chosen through Latin hypercube sampling of the parameters: Mach number; free stream density; free stream temperature; and surface temperature amplitudes, $T_i$.

The mean and max percent errors as well as the overall maximum dimensional errors relative to full-order CFD test case are computed using Eqs. (3.22) to (3.25). The results for laminar flow are listed in Table 4.6. Note that in addition to the surrogate models discussed, the analytical Chapman and Rubesin method is also included using Eqs. (4.14) and (4.20). The “CIM” model produced the lowest errors relative to the 100 test cases, with a mean percent error of 6.4% and a maximum dimensional error of $0.30 \, W/cm^2$ for a test case with a peak heat flux of $3.70 \, W/cm^2$. A comparison of the different models for this test case is also shown in Fig. 4.9. The temperature profile for this case is shown in Fig. 4.9(a) along with the sixth order polynomial curve fit for the “Poly” model. The heat flux results for the “CIM”, “Linear”, “Iso”, “Int”, and “Poly” models are shown in Fig. 4.9(b).

The “Poly” model performed the worst out of the CFD surrogate models, with mean errors of 70% and maximum errors of $13.2 \, W/cm^2$ relative to a test case with a peak heat flux of $4.51 \, W/cm^2$. It was observed that the $b_i$ coefficients for many of the test cases exceed the bounds for which the model was designed (Table 4.5) as shown in Fig. 4.9, significantly degrading the model quality. This highlights a second disadvantage of this approach, in that either $a$ priori knowledge of the surface temperatures are needed, which is not known without carrying out a coupled fluid-thermal-structural analysis, or large initial bounds must best specified. Thus,
Table 4.6: Error in the laminar aerodynamic heat flux models for the flat generic panel compared to CFD.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error ($\text{W/cm}^2$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>39.0</td>
<td>270</td>
<td>1.63</td>
</tr>
<tr>
<td>Iso</td>
<td>35.3</td>
<td>259</td>
<td>1.30</td>
</tr>
<tr>
<td>Int</td>
<td>17.4</td>
<td>110</td>
<td>1.00</td>
</tr>
<tr>
<td>CIM</td>
<td>6.37</td>
<td>47.3</td>
<td>0.30</td>
</tr>
<tr>
<td>Poly</td>
<td>70.3</td>
<td>398</td>
<td>13.2</td>
</tr>
<tr>
<td>Chapman–n = 2</td>
<td>30.1</td>
<td>230</td>
<td>1.26</td>
</tr>
<tr>
<td>Chapman–n = 4</td>
<td>28.9</td>
<td>192</td>
<td>1.08</td>
</tr>
<tr>
<td>Chapman–n = 8</td>
<td>553</td>
<td>15,900</td>
<td>76.7</td>
</tr>
<tr>
<td>Chapman–Iso</td>
<td>36.3</td>
<td>252</td>
<td>1.35</td>
</tr>
</tbody>
</table>

Figure 4.9: Comparison of several laminar aerodynamic heating predictions to the benchmark (CFD) for the flat generic panel: $M_\infty = 5.181$, $\rho_\infty = 196.8 \text{ g/m}^3$, and $T_\infty = 325.3 \text{ K}$.
without a priori understanding of the surface temperature profiles, accuracy can only be ensured through a large sampling of the parameter space, which is computationally expensive relative to the other approaches considered in this study.

The “Linear” and “Iso” models performed similarly, with mean errors above 30% and maximum errors of 1.6 and 1.3 W/cm² relative to test cases with peak heat flux values of 4.14 and 3.70 W/cm², respectively. The “Int” model yielded a mean error of 17% and maximum error of 1.0 W/cm² for to a test case with a peak heat flux of 4.13 W/cm². For a comparison, the heat flux was also predicted using the “Int” model by replacing the surrogate for the heat transfer coefficient with the analytical heat transfer coefficient in Eq. (4.7) was also used to predict the heat flux. This yielded worse errors, namely 62% mean error and 2.35 W/cm² maximum dimensional error for a test case with a peak heat flux of 3.70 W/cm².

Results from the Chapman and Rubesin predictions were at best on par with the “Linear”, “Iso”, and “Int” models. However, setting \( n = 8 \) in Eq. (4.14) yielded the worst error of any cases considered. This is further examined in Fig. 4.10, which are predictions corresponding with the highest error in the “Chapman–Iso” model. Note that increasing values of \( n \) yield improved representation of the temperature profiles, as shown in Fig. 4.10a). However, this also corresponds with diverging heat flux predictions, as shown in Fig. 4.10b), which significantly increases the error of the model.
4.3.3 Comparison of Aerodynamic Heating Models for Turbulent Flow over a Flat Surface

Similar to the laminar flow analysis, 100 CFD evaluation cases with variable free stream properties and spatially varying surface temperatures are computed for turbulent flow over the generic panel using the Menter k-ω SST turbulence model.

The errors relative to 100 CFD test cases are shown in Table 4.7. Again, the “CIM” outperforms all of the other models with a mean error of 2.5% and maximum error 0.88 W/cm² relative to a test case with a peak flux of 31.9 W/cm². The “Linear” and “Poly” models have similar errors relative to the laminar flow results. The “Iso” surrogate has much lower mean and maximum percent errors, but an increased overall dimensional error relative to laminar flow. The “Int” model shows an increase in all three error metrics relative to laminar flow. Note again, for comparison with the surrogate “Int” model, the turbulent analytical heat transfer coefficient
from Eq. (4.8) is also considered for computing the aerodynamic heat flux. The resulting mean and maximum errors are much higher than the surrogate model: mean error of 50.2% and maximum dimensional error of $33.4 \text{ W/cm}^2$ relative to a test case with a peak heat flux of $56.8 \text{ W/cm}^2$. For visual comparison, the results from the models are shown in Fig. 4.11, representing the test case in which the “Iso” model performed the worst. The “CIM” prediction shows excellent agreement with the CFD prediction, while the “Linear” and “Iso” models remain within 3.0 $\text{ W/cm}^2$ of the CFD solution. The “Int” and “Poly” models have upwards of 10.0 $\text{ W/cm}^2$ error for this case.

Table 4.7: Error in the turbulent aerodynamic heat flux models for the flat generic panel compared to CFD.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error ($\text{ W/cm}^2$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>13.8</td>
<td>59.5</td>
<td>7.5</td>
</tr>
<tr>
<td>Iso</td>
<td>7.43</td>
<td>58.3</td>
<td>2.44</td>
</tr>
<tr>
<td>Int</td>
<td>25.8</td>
<td>210</td>
<td>11.2</td>
</tr>
<tr>
<td>CIM</td>
<td>2.53</td>
<td>20.5</td>
<td>0.88</td>
</tr>
<tr>
<td>Poly</td>
<td>29.9</td>
<td>311</td>
<td>56.5</td>
</tr>
</tbody>
</table>

4.3.4 Construction and Verification of CFD Surrogates with Deformation

Aerodynamic heating with variable surface temperature and deformation is also considered for the generic panel. In this analysis only turbulent flow is considered,
(a) Temperature profile and “Poly” profile:
\[ b_0 - b_6 = 590.2, 170.6, -2071, 14981, -43851, 50661, -19995. \]

(b) Heat flux.

Figure 4.11: Comparison of several turbulent aerodynamic heating predictions to the Benchmark (CFD) for the flat generic panel: \( M_\infty = 3.278, \rho_\infty = 334.1 \text{ g/m}^3, T_\infty = 314.5 \text{ K}. \)

using the Menter k-\( \omega \) turbulence model. The flow is held constant at a Mach number of 3.0, and free stream density and free stream temperature equal to the values at a 24 km altitude. The models considered in this analysis are the: “Linear”, “Iso”, “Int”, and “CIM” models. Note the “Poly” model is not considered due to the poor performance in the previous analysis.

Linearized and Integral Models

The construction of the “Linear” model, which neglects the effect of surface temperature and surface deformation on the heat transfer coefficient, is simple for this case. Only two flow solutions are necessary, the aerodynamic heat flux for the surface temperature set to free stream and the adiabatic surface temperature. For this analysis, the adiabatic surface temperature and the heat transfer coeffi-
cient are computed exactly, thus kriging surrogates are unnecessary. Likewise, the “Int” model neglects surface deformation because this adiabatic surface temperature and heat transfer coefficient are used in Eq. (4.10).

**Pointwise Model**

The “Iso” model is constructed from 100 sample points using kriging and LHS with the parameters: (1) isothermal surface temperature, and (2–7) the $a_1$–$a_6$ modal amplitudes for deformation, with the bounds listed in Table 2.1. The mean error for the “Iso” model compared to 25 CFD isothermal solutions with deformation is 3.34%. For comparison, note that the pointwise model without deformation predicts a mean error of 28.8% relative to these test cases which include deformation.

**Corrected Pointwise Model**

The “CIM” is constructed by applying the correction in Eq. (4.32) to the “Iso” model in order to account for non-isothermal surface temperatures. Note, the effects of deformation are assumed to be small in the correction for this configuration, and thus free stream properties are used in place of the boundary layer edge properties in Eq. (4.33). As before, no additional surrogate verification is necessary for this model.
4.3.5 Comparison of Aerodynamic Heating Models with Deformation

In order to compare all of these different models, 100 CFD test cases are computed for turbulent flow with variable: surface temperature and deformation. The test cases are chosen through LHS sampling of the input parameters of: (1–9) surface temperature profile function amplitudes, and (10–15) surface deformation modal amplitudes, \( a_1 – a_6 \). The bounds are the same as those listed previously in Table 2.1.

The mean and max percent errors as well as the overall max dimensional error are computed from Eqs. (3.22) to (3.25). The results are shown in Table 4.8. Note, the “Iso” and “CIM” models which neglect deformation are shown in the table as well, denoted as “Iso–Flat” and “CIM–Flat”, respectively. The model with the lowest errors is the “CIM” with a mean percent error of 2.2%. The worst model is the “Int” model; mean and max errors of 32% and 208%. The “Linear” and “Iso” models have similar levels of error. Neglecting deformation led to modest increases in the error for the “Iso–Flat” model, but approximately 5 times higher error for the “CIM–Flat”.

The test case with the maximum dimensional error for the “CIM” is shown in Fig. 4.12. The temperature profile for this case is shown in Fig. 4.12(a) and the deformation profile is shown in Fig. 4.12(b). For this particular test case, both the “Linear” and “Iso” models have relatively low maximum errors, below 0.3 W/cm². The “Int” model clearly over corrects the influence of the changing surface tem-
Table 4.8: Error in the aerodynamic heat flux for the generic panel with deformation compared to CFD.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error ($W/cm^2$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>24.8</td>
<td>153</td>
<td>0.97</td>
</tr>
<tr>
<td>Iso</td>
<td>16.1</td>
<td>153</td>
<td>0.61</td>
</tr>
<tr>
<td>Int</td>
<td>31.9</td>
<td>208</td>
<td>1.14</td>
</tr>
<tr>
<td>CIM</td>
<td><strong>2.23</strong></td>
<td><strong>18.2</strong></td>
<td><strong>0.12</strong></td>
</tr>
<tr>
<td>Iso–Flat</td>
<td>20.4</td>
<td>134</td>
<td>1.00</td>
</tr>
<tr>
<td>CIM–Flat</td>
<td>11.1</td>
<td>57.1</td>
<td>0.66</td>
</tr>
</tbody>
</table>

Temperature on the heat flux. Finally, though this is the case with the largest error for the “CIM”, it still clearly outperforms the other models, demonstrating that “Iso” portion of the model can capture the effects of deformation, while the correction portion of the model can capture the effects of non-isothermal surface temperatures.
Figure 4.12: Turbulent aerodynamic heat flux comparison of several models compared to the Benchmark (CFD) for the generic panel with deformation. For all of the cases: \( M_\infty = 3.0, \text{Alt.} = 24\text{km}. \)
Chapter 5

Aerodynamic Pressure

Several studies are conducted for the aerodynamic pressure load acting on the surface of hypersonic vehicles. First, similar to the aerodynamic heating results, several turbulence models and an analytical model are compared in order to assess uncertainty in steady-state aerodynamic pressure. Next, the hypothesis of modeling steady-state and unsteady components separately is examined by introducing analytical models for the prediction of the unsteady component of pressure. Then, unsteady Navier-Stokes and Euler solutions are generated for the control surface, and the unsteady components of these solutions are compared to the analytical methods. Model reduction is completed by introducing ROM/S in order to compute the steady-state component of pressure. The steady-state surrogates are then combined with unsteady analytical models to compute the full unsteady aerodynamic pressure.
5.1 Turbulence and Analytical Model Analysis

Similar to aerodynamic heating, different modeling approaches are compared for pressure prediction. The analytical model chosen for this analysis is piston theory, introduced next.

5.1.1 Piston Theory

Classical piston theory [22, 36, 122] provides a simple point-function relationship between the unsteady surface pressure and surface motion. Piston theory uses free stream flow properties as the ambient surface conditions. These properties assume the undeformed surface of interest is parallel to the flow. The derivation of piston theory begins by approximating the normal velocity, \( v_n \), of a slab of fluid, which moves normal to the flow such that,

\[
v_n = \frac{\partial Z(x, y, t)}{\partial t} + U_\infty \left\{ \frac{\partial Z(x, y, t)}{\partial x} \right\} \tag{5.1}
\]

The parameter \( Z(x, y, t) \) defines the surface of the structure of interest, and is given by:

\[
Z(x, y, t) = w(x, y, t) + Z_{str}(x, y) - \alpha(t)x \tag{5.2}
\]

where \( \alpha(t) \) is included to account for free stream flow that is not parallel to the undeformed surface. The equation for the pressure on the surface of a piston which generates only simple waves and has no changes in entropy, is:

\[
\frac{P(x, y, t)}{P_\infty} = \left( 1 + \frac{\gamma - 1}{2} \frac{v_n}{a_\infty} \right)^{\frac{2\gamma}{\gamma-1}} \tag{5.3}
\]
Note that use of Eq. (5.3) assumes no shock is present. Lighthill [36] suggested a third order binomial expansion of Eq. (5.3) in order to account for the effect of shock wave. He found that this expansion yielded pressures within 6% of the simple wave and shock-expansion predictions[36]. Next, the pressure on a moving surface in supersonic/hypersonic flow can be obtained, and in terms of the pressure coefficient [22] is:

\[ C_p(x, y, t) = C_{p,\text{vel}}(x, y, t) + C_{p,S}(x, y, t) + C_p(x, y, t) \]  

(5.4)

where:

\[ C_{p,\text{vel}}(x, y, t) = \frac{2}{M_\infty U_\infty} \left( \frac{\partial Z}{\partial t} \right) + \frac{\gamma + 1}{2} \left( \frac{\partial Z}{\partial t} \right)^2 + \frac{(\gamma + 1)M_\infty}{6U_\infty^3} \left( \frac{\partial Z}{\partial t} \right)^3 \]  

(5.5)

\[ C_{p,S}(x, y, t) = \frac{2}{M_\infty} \left( \frac{\partial Z}{\partial x} \right) + \frac{\gamma + 1}{2} \left( \frac{\partial Z}{\partial x} \right)^2 + \frac{(\gamma + 1)M_\infty}{6} \left( \frac{\partial Z}{\partial x} \right)^3 \]  

(5.6)

\[ \bar{C}_p(x, y, t) = \frac{\gamma + 1}{U_\infty} \left( \frac{\partial Z}{\partial t} \right) \left( \frac{\partial Z}{\partial x} \right) + \frac{(\gamma + 1)M_\infty}{2U_\infty^2} \left( \frac{\partial Z}{\partial t} \right)^2 \left( \frac{\partial Z}{\partial x} \right) \]  

\[ + \frac{(\gamma + 1)M_\infty}{2U_\infty} \left( \frac{\partial Z}{\partial t} \right)^2 \left( \frac{\partial Z}{\partial x} \right) \]  

(5.7)

Note that first-order piston theory is given by the linear terms from Eqs. (5.5) to (5.7), and has been shown to be a poor approximation for unsteady aerodynamics [22]. Accordingly, second-order piston theory corresponds to linear and quadratic terms, and is denoted as “PT²”. Third-order piston theory corresponds to the inclusion of all terms in each equation, and is denoted as “PT³”. Equation (5.5) represents the component of pressure that is entirely dependent on the surface velocity. Equation (5.6) represents the purely steady-state component of pressure, and is dependent on the surface inclination. Equation (5.7) arises because of non-linearity in the flow, and is dependent on the products of surface velocity and inclination.
Note, a transformation of the equations is necessary for three dimensional flow. It is assumed that the surface position of a three-dimensional surface is described by:

\[ Z(x, y, t) = w(x, y, t) + Z_{str}(x, y) - \alpha(t)x \]  

(5.8)

where \( \alpha(t) \) is the rotation of the surface about the mid-chord. For the control surface, \( x \) corresponds to the the coordinate parallel to the root chord of the surface, as illustrated in Fig. 5.1. However, piston theory must be applied in the direction of the free stream flow. Thus, the partial derivatives with respect to \( x \) in Eqs. (5.5) and (5.7) must be replaced by \( x' \), which corresponds to the direction of the free stream. Then, for the surface,

\[ \frac{\partial Z(x, y, t)}{\partial x'} = \frac{\partial Z}{\partial x} \frac{\partial x}{\partial x'} + \frac{\partial Z}{\partial y} \frac{\partial y}{\partial x'} \]  

(5.9)

where for a given side-slip angle, \( B \), Eq. (5.9) simplifies to:

\[ \frac{\partial Z(x, y, t)}{\partial x'} = \cos(B) \frac{\partial Z}{\partial x} + \sin(B) \frac{\partial Z}{\partial y} \]  

(5.10)
Figure 5.1: Direction of the free stream relative to a 3-D structure.
5.1.2 Comparison of Steady-State Pressure Predictions

Several different modeling approaches are compared for steady-state pressure prediction, including: 1) third-order piston theory, 2) RANS with Menter $k - \omega$ SST turbulence model, 3) RANS with Wilcox $k - \omega$ turbulence model, and 4) RANS with Baldwin–Lomax turbulence model. Similar to the aerodynamic heating model comparisons, the HSV panel configuration, the same temperature profile, and the same three panel deformations are considered here.

The temperature profile is shown in Fig. 5.2. The first deformation profile, shown in Fig. 5.3(a), approximates a panel deformation that is influenced by both thermal buckling and aerodynamic pressure [21]. The second, shown in Fig. 5.3(c), consists of deformation in the first sine mode and approximates a panel deformation that is primarily driven by thermal buckling [21]. The third, shown in Fig. 5.3(e), consists of deformation in the third sine mode, which corresponds to panel deformations during post flutter limit cycle oscillations [21].

The aerodynamic pressure solutions for these deformations are shown in Figs. 5.3(b,d,f), respectively. Maximum and average differences between the different models and the RANS with Menter $k - \omega$ SST turbulence model are listed in Table 5.1. Several general conclusions can be made from these results. First, it is clear from Fig. 5.3(d,f) that the largest errors occur at the leading edge of the panel. These errors correlate with the degree of surface inclination at the leading edge between the three different panel shapes. In addition, for all three shapes, piston the-
Figure 5.2: Representative temperature profile used to compare several analytical and turbulence model aerodynamic pressure predictions for the HSV panel.

ory under predicts the pressure relative to the CFD solutions in panel regions with negative slope. Also note that there is a shift in the peak pressure location between the CFD and piston theory solutions. Each of these discrepancies between piston theory and the CFD approaches are likely due to the boundary layer displacement effect not accounted for in piston theory, which is an inviscid model. In contrast to the aerodynamic heating, variation in the aerodynamic pressure predictions for each turbulence model were relatively minor ($< 8\%$ average difference).

Table 5.1: Comparison of steady aerodynamic pressure predictions using different modeling approaches for the HSV panel.

<table>
<thead>
<tr>
<th></th>
<th>Piston Theory</th>
<th>Wilcox Menter</th>
<th>Baldwin-Lomax Menter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max.</td>
<td>76.6%</td>
<td>5.99</td>
<td>18.78</td>
</tr>
<tr>
<td>Avg.</td>
<td>25.7%</td>
<td>1.64</td>
<td>7.65</td>
</tr>
</tbody>
</table>
(a) Panel shape 1, \(a_1^* = 8.8187, a_2^* = -3.1569, a_3^* = 0.1291\).

(b) Aerodynamic pressure for panel shape 1.

(c) Panel shape 2, \(a_1^* = 10.5\).

(d) Aerodynamic pressure for panel shape 2.

(e) Panel shape 3, \(a_1^* = 3.5\).

(f) Aerodynamic pressure for panel shape 3.

Figure 5.3: Comparison of aerodynamic pressure predictions for several HSV panel shapes, \(M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa\).
5.2 Analytical Modeling of Unsteady Pressure

As stated previously, this study seeks to examine model reduction of CFD aerothermodynamics using separate steady-state and unsteady approximations of the flow. As a result of the high flow speeds, the primary component of unsteady pressure in hypersonic flow is due to the surface motion imparting a wash velocity. Because of a slow thermal response time relative to the fluid and structural systems, this unsteadiness is negligible for the heat transfer problem [21]. Thus, no unsteady correction is considered for aerodynamic heating. However, this unsteadiness is known to be important for fluid-structural interactions [22]. Therefore, several approximations are considered to account for this effect. These approximations are based on classical piston theory and local piston theory. The steady and unsteady pressure components are combined to compute the coefficient of pressure according to:

\[ C_p(x, y, t) = C_{p,SS}(x, y, t) + C_{p,US}(x, y, t) \]  

(5.11)

where \( C_{p,SS} \) is the steady-state component of pressure computed from CFD or a CFD-based surrogate and \( C_{p,US} \) is the unsteady component of pressure.

5.2.1 Classical Piston Theory Unsteady Component of Pressure

In order to compute the unsteady component of pressure for piston theory, first consider the classical piston theory equations, (5.5), (5.6), and (5.7). Based on these expressions, the unsteady component of the surface pressure for piston theory is
fairly straightforward,

\[ C_{p,US}(x, y, t) = C_{p,vel}(x, y, t) + \overline{C}_p(x, y, t) \]  \hspace{1cm} (5.12)

where \( C_{p,vel} \) is computed from Eq. (5.5), and \( \overline{C}_p \) is obtained from Eq. (5.7).

### 5.2.2 Local Piston Theory Unsteady Component of Pressure

For problems that exhibit shock-impingement, significant three-dimensional flow, and/or high Mach numbers/surface inclinations, classical piston theory can be modified to local piston theory \([22, 84, 85]\). The most common derivation \([22]\) is completed by replacing free stream flow quantities in Eqs. (5.4) to (5.7) with locally computed flow quantities from a steady-state flow analysis using CFD or CFD-based surrogates. Thus, the structural profile, \( Z \) Eq. (5.8), reduces to only the deformation component \( w(x, y, t) \), since the flow is locally parallel to the surface after the steady-state flow analysis.

An alternative derivation for local piston theory can be obtained by changing the free stream properties in the original piston equation, Eq. (5.3), to local properties and then performing the binomial expansion. Thus:

\[ \frac{P(x, y, t)}{P_{loc(x, y, t)}} = \left( 1 + \frac{\dot{w}(x, y, t)}{a_{loc(x, y, t)}} \right)^{\frac{2\gamma}{\gamma-1}} \]  \hspace{1cm} (5.13)

where, after expansion and neglecting terms higher than first order,

\[ P(x, y, t) = P_{loc(x, y, t)} \left( 1 + \gamma \frac{\dot{w}(x, y, t)}{a_{loc(x, y, t)}} \right) \]  \hspace{1cm} (5.14)

where,

\[ a_{loc(x, y, t)} = \sqrt{\gamma RT_{loc(x, y, t)}} \]  \hspace{1cm} (5.15)
This expression for local piston theory requires two important flow properties. The first is the steady-state local pressure, \( P_{\text{loc}}(x, y, t) \). The second flow property necessary is the local temperature, \( T_{\text{loc}}(x, y, t) \). In viscous flow, the local pressure is assumed to be constant through the boundary layer. Thus, \( P_{\text{loc}}(x, y, t) \) can be evaluated at the surface. However, the local temperature is not constant through the boundary layer, and thus there is some ambiguity as to where this property should be obtained. Several methods are considered here for the evaluation of the local temperature, including: (1) approximating the boundary layer edge in the direction normal to the surface and obtaining \( T_{\text{loc}}(x, y, t) \) at that location, denoted "LPT\(_{\text{BL}}"\), (2) computing steady-state inviscid Euler CFD solutions and obtaining \( T_{\text{loc}}(x, y, t) \) at the surface, denoted "LPT\(_{\text{EU}}"\), and (3) assuming \( T_{\text{loc}}(x, y, t) \) is equal to the free stream \( T_\infty \), denoted "LPT\(_{T_\infty}\)".

For the first method, "LPT\(_{\text{BL}}"\), steady-state viscous CFD flow data is required at the location of the boundary layer edge. This location is approximated by the boundary layer thickness (\( \delta \)), which for laminar flow is [123],

\[
\delta(x) = \frac{5.0x}{\sqrt{Re_x}}
\]  \hspace{1cm} (5.16)

and for turbulent flow [123],

\[
\delta(x) = \frac{0.37x}{Re_x^{1/5}}
\]  \hspace{1cm} (5.17)

Note, these relations are for semi-infinite flat surfaces. However, in the presence of deformation, the boundary layer edge properties would be obtained at \( w(x, y, t) \) plus \( \delta(x) \).
5.3 Unsteady Aerodynamic Pressure Modeling

Several studies are conducted for the prediction of unsteady aerodynamic pressure in high speed flow. The configuration chosen for this analysis is the control surface described in Section 2.5. These studies include inviscid, viscous, and surrogate based models. Assessment of the models is conducted for variations of frequency, mode of oscillation, and for variations of the parameter space defined in Table 2.8. Separation of unsteady pressure into steady-state and unsteady components is examined for both inviscid and viscous flow-fields. Analytical corrections from piston theory and local piston theory are compared to the unsteady components from CFD, in order to assess the accuracy of the analytical methods. Then, surrogates are created to compute the steady-state component of pressure. Finally, the surrogates are combined with the analytical unsteady pressure methods, and the models are compared with full CFD Navier-Stokes solutions. Verification of the models is carried out by a comparison of the generalized aerodynamic forces (GAFs), defined as:

\[
GAF(t) = \frac{1}{2} \rho_\infty U_\infty^2 \int \int \left( \Phi(x, y) C_p(x, y, t) \right) dx dy
\]

(5.18)

where \( \Phi(x, y) \) is the modeshape of the deformation.

5.3.1 Inviscid Results

First, inviscid methods are considered for the control surface. A range of frequencies are included, from 1 Hz up to 200 Hz for both the first and second modes, with
amplitudes: $a_1^* = 1.0$ and $a_2^* = 0.5$. The free stream flow properties are: $M_\infty = 8.0$ and an altitude of 40 km. The steady-state component is approximated for the Euler results by computing Euler CFD solutions using a low frequency motion of the surface, 0.1 Hz, in order to remove the unsteady effect of surface velocity on the pressure.

Several CFD, analytical, and reduced models are considered. The CFD models are: full-order unsteady Euler (“EU”) and full-order steady-state Euler (“EUSS”). The analytical models are: second-order piston theory (“PT²”) and third-order piston theory (“PT³”). The reduced models combine the Euler steady-state solutions with unsteady corrections. These models include: second- and third-order piston theory corrections (“EUSS–PT²” and “EUSS–PT³”), and local piston theory corrections. The local piston theory corrections require the calculation of the local speed of sound, which is computed using: (1) $T_{loc}$ on the surface of steady-state Euler solutions (“EUSS–LPT_EU”) and $T_{loc}$ equal to the free stream $T_\infty$ (“EUSS–LPT_T∞”).

The comparisons are separated into two figures, Figs. 5.4 and 5.5. Figure 5.4 presents the full GAFs for the “EU”, “EUSS”, “PT²”, and “PT³” models for the first mode in (a) and the second mode in (c), and the unsteady components in (b) and (d). Figure 5.5 only presents the unsteady components for the GAFs for the local piston theory models for mode 1 in: (a), and (c); and mode 2 in: (b), and (d). The unsteady components are solely presented for these models, since the only variation between them and the Euler solution (“EU”) is the unsteady component of pressure (i.e., the steady-state component of pressure for all of these models
is “EUSS”). Note, the steady-state Euler model corrected with second- and third-order piston theory is not presented explicitly in these figures, however the unsteady component for these models is exactly the same as for the classical second- and third-order piston theory models shown in Figs. 5.4 (b) and (d).

Figure 5.4: Euler, steady-state Euler, and piston theory Generalized Aerodynamic Forces (GAFs) for oscillations of the control surface. \( f = 1, 25, 50, 100, 150, 200 \ Hz, M_{\infty} = 8.0, \text{Alt.} = 40 \ km. \)

The unsteady components of the GAFs for both modes increase with increasing frequency. The actual natural frequencies of these modes are approximately 13 Hz and 38 Hz, respectively. From these results, it is clear that piston theory and local
Figure 5.5: Unsteady components of the GAFs for the Euler and Euler with local piston theory for oscillations of the control surface. \( a_1^* = 1.0, \ a_2^* = 0.5, \ f = 1, 25, 50, 100, 150, 200 \ Hz, \ M_\infty = 8.0, \ Alt. = 40 \ km. \)
piston theory provide good to excellent approximations for the unsteady components of pressure relative to inviscid Euler aerodynamics, even at frequencies an order of magnitude higher than the natural frequencies of the control surface. Errors in the GAFs are computed relative to the full GAFs for the “EU” model in Table 5.2, using Eqs. (3.22) to (3.25). Most of these models have less than 5.0% mean error, the exception being the steady-state Euler model (“EU_{SS}”). The local piston theory model which evaluated the local temperature at the surface of steady-state Euler solutions (“EU_{SS}–LPT_{EU}”) had the best performance of all of the models considered, with mean errors under 3%.

Table 5.2: Error in GAFs for oscillations of the control surface for several methods compared to Euler CFD solutions.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EU_{SS}</td>
<td>54.1</td>
<td>142</td>
<td>67.0</td>
</tr>
<tr>
<td>PT^2</td>
<td>8.36</td>
<td>43.4</td>
<td>12.8</td>
</tr>
<tr>
<td>PT^3</td>
<td>8.40</td>
<td>37.4</td>
<td>4.94</td>
</tr>
<tr>
<td>EU_{SS}–PT^2</td>
<td>3.41</td>
<td>18.1</td>
<td>9.15</td>
</tr>
<tr>
<td>EU_{SS}–PT^3</td>
<td>4.41</td>
<td>9.56</td>
<td>4.58</td>
</tr>
<tr>
<td>EU_{SS}–LPT_{EU}</td>
<td><strong>2.69</strong></td>
<td><strong>5.89</strong></td>
<td><strong>2.92</strong></td>
</tr>
<tr>
<td>EU_{SS}–LPT_{T∞}</td>
<td>4.70</td>
<td>13.7</td>
<td>6.94</td>
</tr>
</tbody>
</table>

5.3.2 Viscous Results

Viscous results are compared for the same two cases as in the inviscid analysis. However, the surface temperature is necessary for the computation of viscous flow
solutions. Thus, the first result considered is a comparison of Navier-Stokes ("NS") results for two different surface temperatures, free stream (262 K) and 1200 K. This result, along with the Euler ("EU") result is shown in Fig. 5.6 for modes one and two. It is clear from this analysis, that the almost 1000 K surface temperature difference between the two Navier-Stokes solutions creates negligible differences in the GAFs, 3.6% mean. Therefore, for this three-dimensional pressure analysis, surface temperature plays only a minor role.

Figure 5.6: Navier-Stokes and Euler Generalized Aerodynamic Forces (GAFs) for surface temperatures of free stream (262 K) and 1200 K for oscillations of the control surface. $f = 1, 25, 50, 100, 150, 200$ Hz, $M_\infty = 8.0$, Alt. = 40 km.
Similar to the inviscid analysis, several full-order CFD, analytical, and reduced models are considered for the viscous analysis. The “NS” result with a surface temperature of 1200 K is considered, along with a steady-state CFD Navier-Stokes (“NSSS”) solution using the same surface temperature and computed using low frequency motion, 0.1 Hz. The analytical models included are: second-order (“PT2”) and third-order (“PT3”) piston theory. The reduced models included combine the “NSSS” solutions with unsteady corrections from both of the piston theory models and the three local piston theory models. Note, this analysis includes the local piston theory correction which evaluates $T_{loc}$ using boundary layer edge properties from the steady-state Navier-Stokes solution (“NS–LPTBL”). The boundary layer edge is approximated a priori using the boundary layer thickness equation, Eq. (5.17). The errors relative to the 1200 K “NS” result for all of the models are listed in Table 5.3. The corrected steady-state Navier-Stokes model with second-order piston theory has the lowest mean and max percent errors, while classical second-order piston theory has the overall lowest dimensional error. Due to a poor approximation of the boundary layer edge, the local piston theory models performed poorly for this analysis.

For visual comparison of the GAFs, only the unsteady component of the models is considered. By definition the unsteady pressure component of the Navier-Stokes result is simply the full pressure minus the steady-state pressure, thus the accuracy of the correction methods relative to the unsteady component of the GAFs will demonstrate the overall accuracy of the models. The classical piston theory
Table 5.3: Error in GAFs for oscillations of the control surface for several methods compared to the 1200 K Navier-Stokes CFD solutions.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Diff. (%)</th>
<th>Max Diff. (%)</th>
<th>Max Diff. (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>NS (T_w = 262 K)</td>
<td>3.61</td>
<td>10.61</td>
<td>1.81</td>
</tr>
<tr>
<td>NS_SS</td>
<td>51.3</td>
<td>138</td>
<td>60.5</td>
</tr>
<tr>
<td>EU</td>
<td>15.6</td>
<td>32.1</td>
<td>9.08</td>
</tr>
<tr>
<td>PT^2</td>
<td>9.90</td>
<td>22.8</td>
<td>6.49</td>
</tr>
<tr>
<td>PT^3</td>
<td>19.6</td>
<td>60.7</td>
<td>12.8</td>
</tr>
<tr>
<td>NS_SS–PT^2</td>
<td>3.83</td>
<td>15.0</td>
<td>7.03</td>
</tr>
<tr>
<td>NS_SS–PT^3</td>
<td>7.80</td>
<td>23.1</td>
<td>10.8</td>
</tr>
<tr>
<td>NS_SS–LPT_{BL}</td>
<td>19.3</td>
<td>60.4</td>
<td>21.7</td>
</tr>
<tr>
<td>NS_SS–LPT_{EU}</td>
<td>17.0</td>
<td>48.3</td>
<td>22.7</td>
</tr>
<tr>
<td>NS_SS–LPT_{T∞}</td>
<td>19.3</td>
<td>52.6</td>
<td>24.7</td>
</tr>
</tbody>
</table>

models and the corrected piston theory models have the exact same unsteady component, and are shown in Figs. 5.7 (a) and (b). For both modes one and two the third-order piston theory unsteady component over-predicts the Navier-Stokes result but predicts roughly the correct profile, while the second-order unsteady component does not predict the correct profile but predicts approximately the correct amplitude. Thus, the second-order piston theory unsteady component is a better approximation of the Navier-Stokes result.

The three local piston theory models are presented in Figs. 5.7 (c) and (e) for mode one, and (d) and (f) for mode two. The local temperature approximated using either steady-state Euler solutions or using the free stream temperature over-
predicts the Navier-Stokes results by approximately 17% for the highest frequency. The local piston theory with the local temperature approximated edge of the boundary layer under-predicts the Navier-Stokes results by 19%.

Another important item to highlight is the increased accuracy of the corrected steady-state model over the basic steady-state model, where the only difference between these models is the use of piston theory-based corrections for surface wash velocity. Thus, despite the high speed nature of the flow, the impact of the wash velocity on the generalized aerodynamic force is non-negligible. This is evident through a visual comparison of the relative magnitude difference between the unsteady and steady components of the GAFs for the two example cases shown in Fig. 5.6.
Figure 5.7: Unsteady components of the GAFs for Navier-Stokes, piston theory, and local piston theory for oscillations of the control surface. $a_1^* = 1.0$, $a_2^* = 0.5$, $f = 1, 25, 50, 100, 150, 200$ Hz, $M_\infty = 8.0$, Alt. = 40 km.
5.3.3 Surrogate Results

In order to complete the model reduction analysis for the control surface, a steady-state surrogate models are constructed. Several different methods are possible for computing the steady-state pressure using a surrogate model of CFD; including: (1) the linear approach, which neglects deformation and thus only accounts for the initial geometry, (2) the pointwise approach, and (3) the polynomial surface temperature basis approach. Note, no corrections are included to account for surface temperature gradients on the aerodynamic pressure, due to the weak coupling between surface pressure and surface temperature [23]. For the demonstration of surrogate modeling for the control surface, only one method is included in this analysis: the “Poly” temperature basis model. The other methods are considered in the fluid-thermal-structural analysis chapters.

The polynomial surface temperature basis for the control surface is given in Eq. (3.21). The bounds of the polynomial are arbitrarily determined a priori as: \( b_0 = 200 \) to \( 1500 \) K, and \( b_1 - b_5 = \pm 650 \). The remaining parameters and bounds are listed in Table 2.8, for a total of 12 input parameters. A kriging surrogate is constructed using LHS to pick 1200 sample points, and then the steady-state Navier-Stokes training solutions are generated. Due to the general poor performance of the local piston theory methods in the previous analysis, only second- and third-order corrections are considered for the “Poly” model for the control surface. The errors relative to the 1200 K “NS” result are listed in Table 5.4. Note, these errors are simi-
lar in magnitude to the steady-state Navier-Stokes model, thus the “Poly” model is capable of accurately predicting the steady-state Navier-Stokes component of the GAFs.

Table 5.4: Error in GAFs for oscillations of the control surface for surrogate models compared to the 1200 K Navier-Stokes CFD solutions.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Poly</td>
<td>77.6</td>
<td>168</td>
<td>66.8</td>
</tr>
<tr>
<td>Poly–PT(^2)</td>
<td>4.29</td>
<td>16.0</td>
<td>7.53</td>
</tr>
<tr>
<td>Poly–PT(^3)</td>
<td>8.03</td>
<td>22.9</td>
<td>10.8</td>
</tr>
</tbody>
</table>

For visual comparison of the GAFs, the surrogate model with both piston theory corrections is shown in Fig. 5.8. The GAFs for the second-order correction are shown in figures (a) and (b) for modes one and two, while the GAFs third-order correction are shown in (c) and (d). From these results, it is clear that even up to a frequency of 200 Hz, the surrogate with second-order piston theory correction (“Poly–PT\(^2\”) is highly accurate in representing the unsteady aerodynamic pressure of an oscillating three-dimensional surface. It is also clear that this model provides the best approximation for these cases, aside from the second-order piston theory corrected steady-state Navier-Stokes model. The increased error of the Euler prediction compared to the surrogate is due to inviscid-viscous interactions in hypersonic flow, which results in a boundary layer displacement effects and modifies the surface pressure. The good accuracy of the surrogate suggests that these effects primarily impact the steady component of the GAFs, and that they can be
captured accurately using a steady-state surrogate even in three-dimensional flow.

Figure 5.8: Navier-Stokes and surrogate Generalized Aerodynamic Forces (GAFs) for the control surface. $f = 1, 25, 50, 100, 150, 200$ Hz, $M_{\infty} = 8.0$, Alt. = 40 km.

Next, an expanded verification of the second-order piston theory corrected surrogate is performed over the parameter space listed in Table 2.8. Comparisons are made to Navier-Stokes, Euler, and classical piston theory aerodynamics; using three different deformation modes and five different combinations of input parameters, providing 15 different comparison cases. The first deformation mode considered is the first bending mode oscillating at its natural frequency of 13.41 Hz.
The second deformation mode considered is the second free vibration mode oscillating at its natural frequency of 37.51 Hz. The third deformation mode is a linear combination of the first two, oscillating at the average frequency of the two modal frequencies, i.e., 25.46 Hz. A summary of the different cases is provided in Table 5.5; where the altitude and surface temperature are held constant at 40 km and 1200 K respectively. Note that a reduced parameter space relative to Table 2.8 was used due to the difficulty to obtain unsteady Navier-Stokes and Euler CFD solutions in the extreme sets of conditions.

Table 5.5: Parameter space for control surface unsteady comparison cases.

<table>
<thead>
<tr>
<th>$7.0 \leq M_\infty \leq 9.0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$-4.0^\circ \leq \alpha \leq 4.0^\circ$</td>
</tr>
<tr>
<td>$-3.0^\circ \leq \beta \leq 3.0^\circ$</td>
</tr>
<tr>
<td>$-1.0 \leq a_1^* \leq 1.0$</td>
</tr>
<tr>
<td>$-0.5 \leq a_2^* \leq 0.5$</td>
</tr>
</tbody>
</table>

Example results are shown in Fig. 5.9, and the error for all of the 15 cases is listed in Table 5.6. Several interesting items can be noted from these results. First, similar to the previous results, the second-order piston theory corrected surrogate yields the lowest mean and maximum error relative to Navier-Stokes. Second, the unsteady Euler model provides the best accuracy outside of the corrected surrogate, outperforming the classical piston theory methods. However, the unsteady component of the Euler GAFs have a slightly wider band of error compared to the
second-order piston theory correction - producing both better and worse results for different cases. Additionally, third-order piston theory produces results much closer to the Euler solution than does second-order piston theory, particularly evident in Figs. 5.9 (c), (d), and (f). Also, it is interesting that the transient shape of the Euler and third-order piston theory unsteady components more closely resembles the shape of the Navier-Stokes result, however the amplitude of the second-order piston theory correction makes it a better approximation, clearly seen in (f).

Table 5.6: Error in GAFs for the control surface for the Navier-Stokes solution with 1200 K surface temperature compared to several models over the parameter space.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>NSSS</td>
<td>13.3</td>
<td>20.7</td>
<td>13.0</td>
</tr>
<tr>
<td>EU</td>
<td>7.95</td>
<td>14.3</td>
<td>9.02</td>
</tr>
<tr>
<td>PT²</td>
<td>13.4</td>
<td>28.6</td>
<td>39.0</td>
</tr>
<tr>
<td>PT³</td>
<td>12.5</td>
<td>25.3</td>
<td>29.6</td>
</tr>
<tr>
<td>Poly</td>
<td>13.1</td>
<td>21.6</td>
<td>14.5</td>
</tr>
<tr>
<td>Poly–PT²</td>
<td><strong>2.61</strong></td>
<td><strong>6.00</strong></td>
<td><strong>4.65</strong></td>
</tr>
<tr>
<td>Poly–PT³</td>
<td>3.74</td>
<td>7.38</td>
<td>4.72</td>
</tr>
</tbody>
</table>
(a) GAFs Case 1: \( M_\infty = 9.0, \alpha = -2^\circ, \beta = 1^\circ, a_1^* = \sin(2\pi ft), a_2^* = 0, f = 13.41 \text{ Hz.} \)

(b) Unsteady Component of GAFs Case 1.

(c) GAFs Case 2: \( M_\infty = 7.0, \alpha = 4^\circ, \beta = 3^\circ, a_1^* = 0, a_2^* = 0.5\sin(2\pi ft), f = 37.51 \text{ Hz.} \)

(d) Unsteady Component of GAFs Case 2.

(e) GAFs Case 3: \( M_\infty = 8.0, \alpha = 1^\circ, \beta = 2^\circ, a_1^* = 0.75\sin(2\pi ft), a_2^* = 0.5\sin(2\pi ft), f = 25.46 \text{ Hz.} \)

(f) Unsteady Component of GAFs Case 3.

Figure 5.9: Navier-Stokes, Euler, piston theory, and surrogate GAFs for the control surface. \( T_w = 1200 \text{ K, Alt.} = 40 \text{ km.} \)
Chapter 6

Fluid-Thermal-Structural Analysis: HSV Panel

In this study, the fluid-thermal-structural analysis of the Hypersonic Vehicle (HSV) panel described in Section 2.2 is considered. The structure is modeled as a simply-supported panel undergoing cylindrical bending. As shown in Fig. 2.3, the panel is assumed to be located on the surface of a two-dimensional wedge, representing the forebody surface of a hypersonic vehicle. The fluid-thermal-structural behavior of such a panel has been studied extensively in Ref. [21] using analytical and semi-empirical aerothermodynamic models.

In order to compute the fluid-thermal-structural responses using CFD-based loads for this configuration, aerodynamic heating and unsteady pressure surrogates are constructed. Then, the fluid-thermal-structural method of solution is introduced, and the results of the simulation are presented.
6.1 Aerodynamic Heat Flux Modeling

The HSV panel discussed in Section 2.2, considers two-dimensional flow over a wedge-type vehicle with variable free stream Mach numbers. The wedge creates an oblique shock, which is modeled using oblique shock theory [87]. The CFD computational domain is based on the flow conditions behind the oblique shock, denoted region 2. Due to the different Mach numbers, the CFD model will have variable inflow densities, temperatures, and velocities. This model also considers variable surface temperature and deformation, using the bounds listed in Table 2.3. In order to determine the error introduced by a surrogate representation of the full-order aerodynamic heating, verification of the surrogates is performed by a comparison with full-order CFD test cases inside the parameter space used to construct the surrogates. Next, in order to determine the error involved in the coupling strategies, the aerodynamic heating surrogates are compared to 100 CFD test cases with non-uniform surface temperatures, and variable surface deformation, and free stream densities, temperatures, and velocities. For this analysis, the Menter $k-\omega$ SST turbulence model is used in all CFD training and evaluation cases.

6.1.1 Construction and Verification of CFD Surrogates

The “Linear”, “Iso”, “CIM”, and “Poly” models are constructed for this analysis.
Linearized Model

The “Linear” model neglects deformation, and is created using a kriging surrogate of the heat transfer coefficient and the adiabatic surface temperature, with input parameters of: (1) $M_{2}$, (2) $\rho_{2}$, and (3) $T_{2}$. Latin Hypercube Sampling is used to pick 250 sample points, and 500 CFD solutions are obtained, with surface temperatures set to free stream and adiabatic. Relative to 25 CFD test cases, the errors for the heat transfer coefficient are a mean of 4.80% and a max of 5.28%, while the errors for the adiabatic surface temperature are a mean of 3.24% and a max of 8.92%.

Pointwise Model

The “Iso” model is created from 1500 sample points using the parameters and bounds in Table 2.3. Note, compared to 25 isothermal test cases with the same bounds, the “Iso” model has a mean error of 2.14% and a maximum error of 13.1%.

Corrected Pointwise Isothermal Model

The “CIM” is constructed using the “Iso” surrogate model in Eq. (4.32). Note, the effects of deformation are assumed to be small in the correction, and thus free stream properties are used in place of the boundary layer edge properties in Eq. (4.33). As before, no additional surrogate verification is necessary for this model.
Parameterization of the Surface Temperature using a Polynomial Basis

Additionally, a heat flux surrogate is computed by parameterizing the surface temperature in terms of the polynomial basis in Eq. (4.13), denoted as the “Poly” model. This model is constructed from 1500 sample points, again using the bounds in Table 2.3. The surface temperature in the sample points is defined in the same manner as for the generic 2-D panel, using the profile in Eq. (4.13) and bounds listed in Table 4.5. The “Poly” model compared to 25 test cases using the same bounds as those in the model, resulted in a mean error of 4.44% and a max error of 44.9%.

6.1.2 Comparison of Aerodynamic Heating Models

In order to compare these different models, 100 CFD test cases are computed for turbulent flow with the variable: surface temperature profiles, deformation, and operating conditions listed in Table 2.3. The test cases are chosen through LHS sampling of the input parameters: (1–9) surface temperature profile function amplitudes from Section 4.3.2, (10) $M_2$, (11) $\rho_2$, (12) $T_2$, and (13–18) surface deformation modal amplitudes, $a_1^*–a_6^*$.

The mean and max percent errors as well as the overall max dimensional error are computed and the results are shown in Table 6.1. The model with the lowest errors is the “CIM” with a mean percent error of 2.55% and maximum error of 22.7%. The “Iso” model has similar levels of error, indicating that the correction
provides only a small improvement for this configuration. This is presumably due to the simply-supported deformation, which dominates the aerodynamic heating response, relative to the variable surface temperatures considered. The “Linear” model, which neglects deformation entirely, has substantially larger errors. The “Poly” model also performs poorly, with mean and max errors of 59.1%, 320%. This demonstrates that a model which is relatively good inside of the parameter space in which it is designed, mean errors under 5%, can have poor results when extrapolating outside of its designed parameter space, approximately 60% mean error.

Table 6.1: Error in the aerodynamic heat flux for the HSV panel compared to CFD.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (W/cm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>46.2</td>
<td>351</td>
<td>94.2</td>
</tr>
<tr>
<td>Iso</td>
<td>2.57</td>
<td>24.0</td>
<td>3.68</td>
</tr>
<tr>
<td>CIM</td>
<td>2.55</td>
<td>22.7</td>
<td>3.57</td>
</tr>
<tr>
<td>Poly</td>
<td>59.1</td>
<td>320</td>
<td>84.9</td>
</tr>
</tbody>
</table>

The test case with the maximum dimensional error for the “CIM” is shown in Fig. 6.1. The temperature profile for this case is shown in Fig. 6.1(a) along with the sixth order polynomial curve fit for the “Poly” model. The deformation profile is shown in Fig. 6.1(b). For this particular test case, both the “CIM” and “Iso” models have low maximum errors, below 3.7 W/cm², which occur near the trailing edge. As before, the the “CIM” is limited by the accuracy of the “Iso” model relative to the
isothermal solutions, and near the trailing edge the error is clearly due to the “Iso” model, not due to the correction in the “CIM”. The poor results for the “Linear” and “Poly” models are also illustrated, the “Linear” model does not capture the variation in heat flux due to deformation, and the “Poly” model is extrapolating far outside of the bounds of the $b_i$ coefficients for which it was designed.
(a) Temperature Profile and Poly Profile: $b_0 - b_6 = 1305, -1191, 15097, -58325, 99385, -77646, 22812$

(b) Deformation: $a_1^* - a_6^* = -12.07, -0.19, 1.20, 1305, -1191, 15097, -58325, 99385, -77646, -0.42, 1.01, 0.75$

(c) Heat Flux.

Figure 6.1: Turbulent aerodynamic heat flux comparison of several models compared to the Benchmark (CFD) for the HSV panel. For this case: $M_2 = 8.36$, $\rho_2 = 39.35 \text{g/m}^3$, $T_2 = 324.4 \text{K}$. 

138
6.2 Unsteady Aerodynamic Pressure Modeling

Several studies are conducted for the prediction of unsteady aerodynamic pressure on the HSV panel. These studies include inviscid, viscous, and surrogate based models. Verification of the models is carried out by a comparison of the generalized aerodynamic forces (GAFs), defined in Eq. (5.18). First, inviscid results are considered.

6.2.1 Inviscid Results

Inviscid unsteady Euler results are generated for three cases for the HSV panel. The first case includes oscillations of the first mode, the second case considers the second mode, and the third case the third mode. The frequency and operating conditions are held constant between all three cases at 140 Hz, with amplitudes of $a_1^* = 5$, $a_2^* = 2$, and $a_3^* = 1$, respectively. These values are similar to those observed during post-flutter limit cycle oscillations in Ref. [21]. The flow properties used in this analysis are: $M_2 = 6.86$, $T_2 = 300K$, and $P_2 = 2966Pa$. In order to further examine model reduction in terms of steady-state and unsteady components, the steady-state component is computed by repeating the three unsteady results using a low frequency surface motion (0.1 Hz).

Several full order models, analytical models, and reduced models are considered. The full-order models are the unsteady Euler (“EU”) CFD solution and the steady-state CFD Euler (“EUSS”) solution. The analytical models are second-
order piston theory ("PT^2") and third-order piston theory ("PT^3") described in Section 5.1.1. The reduced models combine the Euler steady-state solutions with unsteady corrections. These models are described in Section 5.2, and include: second- and third-order piston theory corrections ("EUSS–PT^2" and "EUSS–PT^3"), and the local piston theory corrections. The local piston theory corrections require the calculation of the local speed of sound, which is computed using: (1) $T_{loc}$ on the surface from the steady-state Euler model ("EUSS–LPT\_EU") and (2) $T_{loc}$ equal to the free stream temperature $T_2$ ("EUSS–LPT\_T_\infty").

The comparisons are separated into three figures all of which include the "EU" and "EUSS" models. Figure 6.2 also contains the classical second- and third-order piston theory models. Figure 6.3 includes the "EUSS" model with second- and third-order piston theory corrections. Figure 6.4 includes the "EUSS" model with local piston theory corrections. Note, for all three figures, (a), (c), and (e) are the full GAFs for modes one to three respectively, while (b), (d), and (f) represent the unsteady components of the GAFs for mode one to three. Note the unsteady component of the Euler-based models is computed by subtracting out the steady-state component of the pressure, the "EUSS" model.

Figures 6.2 to 6.4 (a), (c), and (e) highlight dramatic differences between the "EUSS" model and all of the other models considered, indicating significant unsteady effects. Not surprisingly, the largest unsteady contribution occurs for the case with largest amplitude surface motion, mode 1. The smaller the modal amplitudes, the smaller the unsteady component.
In the cases with oscillations of the second mode, Figs. 6.2 to 6.4 (c) and (d), the steady-state component of the GAFs is approximately 0 for all time. This is due to the equal compression and expansion computed using inviscid flow for the second sinusoidal mode, thus reducing the integrated pressure on the surface to 0, independent of time. Finally, the unsteady components for second- and third-order piston theory and the unsteady component of the corrected steady-state model using second- and third-order piston theory are exactly identical by definition.

Errors in the GAFs are computed relative to the “EU” model in Table 6.2, using Eqs. (3.22) to (3.25). Most of these models perform exceptionally well with less than 2% mean error and 5% maximum percent error; the exceptions being the steady-state Euler model ("EU_{SS}") and the local piston theory correction using the free stream temperature ("EU_{SS}-LPT_{T∞}"). The steady-state Euler corrected with second-order piston theory has the lowest mean and maximum percent errors: 1.1% and 3.6%, while classical second-order piston theory has the lowest overall maximum dimensional error, 2.28 N. These results indicate that for 2-D inviscid flow, separating the full unsteady transient pressure into steady-state components and an unsteady correction component from piston theory or local piston theory is valid for non-dimensional amplitudes and frequencies up to 5.0 and 140 Hz, respectively.
(a) GAFs Mode 1. $a_1^* = 5.0$

(b) Unsteady Component of GAFs, Mode 1.

(c) GAFs Mode 2. $a_2^* = 2.0$

(d) Unsteady Component of GAFs, Mode 2.

(e) GAFs Mode 3. $a_3^* = 1.0$

(f) Unsteady Component of GAFs, Mode 3.

Figure 6.2: Euler and piston theory Generalized Aerodynamic Forces (GAFs) for oscillations of the HSV panel. $f = 140 \text{ Hz}$, $M_2 = 6.86$, $T_2 = 300K$, $P_2 = 2966Pa$. 
Figure 6.3: Euler and piston theory corrected steady-state Euler model GAFs for oscillations of the HSV panel. $f = 140 \text{ Hz}$, $M_2 = 6.86$, $T_2 = 300K$, $P_2 = 2966\text{ Pa}$.
Figure 6.4: Euler and local piston theory corrected steady-state Euler model GAFs for oscillations of the HSV panel. $f = 140 \text{ Hz}, M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa$. 

(a) GAFs Mode 1. $a_1^* = 5.0$

(b) Unsteady Component of GAFs, Mode 1.

(c) GAFs Mode 2. $a_2^* = 2.0$

(d) Unsteady Component of GAFs, Mode 2.

(e) GAFs Mode 3. $a_3^* = 1.0$

(f) Unsteady Component of GAFs, Mode 3.
Table 6.2: Error in GAFs for oscillations of the HSV panel for several methods compared to the Euler CFD solution (EU) over all three modes.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EU&lt;sub&gt;SS&lt;/sub&gt;</td>
<td>84.6</td>
<td>141</td>
<td>203</td>
</tr>
<tr>
<td>PT&lt;sup&gt;2&lt;/sup&gt;</td>
<td>1.30</td>
<td>4.34</td>
<td>2.28</td>
</tr>
<tr>
<td>PT&lt;sup&gt;3&lt;/sup&gt;</td>
<td>1.42</td>
<td>4.69</td>
<td>2.97</td>
</tr>
<tr>
<td>EU&lt;sub&gt;SS&lt;/sub&gt;–PT&lt;sup&gt;2&lt;/sup&gt;</td>
<td><strong>1.13</strong></td>
<td><strong>3.58</strong></td>
<td>2.59</td>
</tr>
<tr>
<td>EU&lt;sub&gt;SS&lt;/sub&gt;–PT&lt;sup&gt;3&lt;/sup&gt;</td>
<td>1.45</td>
<td>3.93</td>
<td>2.74</td>
</tr>
<tr>
<td>EU&lt;sub&gt;SS&lt;/sub&gt;–LPT&lt;sub&gt;EU&lt;/sub&gt;</td>
<td>1.58</td>
<td>5.10</td>
<td>8.14</td>
</tr>
<tr>
<td>EU&lt;sub&gt;SS&lt;/sub&gt;–LPT&lt;sub&gt;T&lt;sub&gt;∞&lt;/sub&gt;&lt;/sub&gt;</td>
<td><strong>14.4</strong></td>
<td>24.1</td>
<td>38.5</td>
</tr>
</tbody>
</table>
6.2.2 Viscous Results

Viscous models are also considered for the same three cases as the inviscid results. However, the surface temperature is necessary as a boundary condition for viscous flow solutions. Thus, the first result considered is a comparison of Navier-Stokes (“NS”) results for two different surface temperatures, 300 K and 900 K. This result, along with the Euler (“EU”) result is shown in Fig. 6.5 for modes one to three. It is clear from this analysis, that the 600 K surface temperature difference in the Navier-Stokes solutions led to small differences, 3.5% mean and 6.9% maximum. Thus surface temperature plays a small role in the unsteady pressure acting on the surface of this panel; compared to the difference between viscous and inviscid modeling: 76% mean and 155% maximum. Note these difference are listed in Table 6.3.

Similar to the inviscid analysis, several full order models, analytical models, and reduced models are considered for the viscous analysis. The “NS” result with a surface temperature of 300 K is used along with a steady-state Navier-Stokes (“NSss”) solution using the same surface temperature, computed using the low frequency of 0.1 Hz. The analytical models considered are second-order (“PT2”) and third-order (“PT3”) piston theory. The reduced models considered combine the “NSss” solutions with unsteady corrections from both piston theory models and the local piston theory models described in the inviscid analysis. Also included is the local piston theory correction which computes $T_{loc}$ using boundary
Figure 6.5: Navier-Stokes and Euler Generalized Aerodynamic Forces (GAFs) for surface temperatures of 300K and 900K for oscillations of the HSV panel. $f = 140 \, Hz$, $M_2 = 6.86$, $T_2 = 300K$, $P_2 = 2966Pa$. 

(a) GAFs Mode 1. $a_1^* = 5.0$

(b) Unsteady Component of GAFs, Mode 1.

(c) GAFs Mode 2. $a_1^* = 2.0$

(d) Unsteady Component of GAFs, Mode 2.

(e) GAFs Mode 3. $a_1^* = 1.0$

(f) Unsteady Component of GAFs, Mode 3.
layer edge properties from a steady-state Navier-Stokes solution ("NS–LPT\textsubscript{BL}").

The boundary layer edge is approximated \textit{a priori} using the boundary layer thickness equation, Eq. (5.17). The errors relative to the 300 K "NS" result for all of the models are listed in Table 6.3.

Table 6.3: Difference in GAFs for the HSV panel for the Navier-Stokes solution with 300 K surface temperature compared to several other methods over all three modes.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Diff. (%)</th>
<th>Max Diff. (%)</th>
<th>Max Diff. (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>NS ($T_w = 900K$)</td>
<td>3.45</td>
<td>6.87</td>
<td>16.8</td>
</tr>
<tr>
<td>NS\textsubscript{SS}</td>
<td>18.9</td>
<td>54.6</td>
<td>185</td>
</tr>
<tr>
<td>EU</td>
<td>76.0</td>
<td>155</td>
<td>376</td>
</tr>
<tr>
<td>PT\textsuperscript{2}</td>
<td>76.2</td>
<td>155</td>
<td>377</td>
</tr>
<tr>
<td>PT\textsuperscript{3}</td>
<td>76.2</td>
<td>155</td>
<td>377</td>
</tr>
<tr>
<td>NS\textsubscript{SS}–PT\textsuperscript{2}</td>
<td>7.58</td>
<td>21.6</td>
<td>73.4</td>
</tr>
<tr>
<td>NS\textsubscript{SS}–PT\textsuperscript{3}</td>
<td>7.71</td>
<td>21.7</td>
<td>73.5</td>
</tr>
<tr>
<td>NS\textsubscript{SS}–LPT\textsubscript{BL}</td>
<td>\textbf{6.79}</td>
<td>\textbf{17.9}</td>
<td>\textbf{60.6}</td>
</tr>
<tr>
<td>NS\textsubscript{SS}–LPT\textsubscript{EU}</td>
<td>9.20</td>
<td>23.5</td>
<td>79.7</td>
</tr>
<tr>
<td>NS\textsubscript{SS}–LPT\textsubscript{T\infty}</td>
<td>13.6</td>
<td>32.8</td>
<td>111</td>
</tr>
</tbody>
</table>

For visual comparison of the GAFs, only a few of the models are considered: “NS”, “EU”, “NS\textsubscript{SS}”, “NS\textsubscript{SS}–PT\textsuperscript{2}”, “NS\textsubscript{SS}–LPT\textsubscript{BL}”. The results from these models are shown in Fig. 6.6. The full GAFs are shown in Figs. 6.6 (a), (c), and (e) for all three modes, while (b), (d), and (f) are the unsteady components of the GAFs. Similar to the inviscid analysis, the unsteady component of the Navier-Stokes based models is computed by subtracting out the steady-state “NS\textsubscript{SS}” pressure. The er-
ror due to inviscid modeling is clearly seen in the difference between the “EU” and “NS” results.

For the first mode, Figs. 6.6 (a) and (b), the “NS_{ss}” model performs poorly due to the large unsteady component of the GAFs. However, for the higher modes, the unsteady component is much smaller in magnitude, and subsequently the “NS_{ss}” model has lower errors. Additionally, for the higher modes, the unsteady component for the “NS” result diverges from the other models considered. This difference between the “NS” result and the other methods is due to boundary layer displacement effects, which are not included in the piston theory corrected models and are only present in scaling the local piston theory corrected models. These effects become more important for the higher modes. However, the unsteady components for these modes are small compared to the full GAFs. Thus, these results indicate that for 2-D viscous flow, full unsteady transient pressure can again be modeled fairly accurately by separately modeling the steady-state components and subsequently adding inviscid unsteady correction components from piston theory or local piston theory.
Figure 6.6: Navier-Stokes, Euler, and corrected steady-state Navier-Stokes Generalized Aerodynamic Forces (GAFs) for oscillations of the HSV panel. \( f = 140 \text{ Hz} \), \( M_2 = 6.86 \), \( T_2 = 300K \), \( P_2 = 2966 Pa \), \( T_w = 300K \), \( T_r = 0.085 \).
6.2.3 Surrogate Results

In order to complete the model reduction analysis for the HSV panel, the steady-state surrogate models are combined with the unsteady correction methods and compared to the fully unsteady Navier Stokes results. Only two surrogate models are included in this analysis, the pointwise surrogate (“Iso”) and the polynomial surface temperature basis surrogate (“Poly”).

Due to the weak coupling between surface temperature and pressure, no correction is considered for the pointwise “Iso” model in order to account for the effect of surface temperature gradients on the pressure. The surrogates are constructed from the same 1500 training snap shots generated for the aerodynamic heating analysis of the HSV panel, described in Section 6.1.

Again for comparison, several full-order models and reduced models are considered for the three cases. The full-order models included are the “NS” and “NSss” methods. The reduced models included combine the steady-state “Iso” and steady-state “Poly” models with unsteady corrections from both piston theory methods and the local piston theory methods. Note, for the surrogates with local piston theory correction “LPT_{BL}”, the local temperature at the edge of the boundary layer is obtained from the 1500 CFD training solutions and is then modeled using the surrogates. The errors relative to the 300 K “NS” result for all of the models are listed in Table 6.4.

For visual comparison of the GAFs, only a few of the models are considered:
Table 6.4: Error of several surrogate GAFs for the HSV panel compared to the Navier-Stokes solution with 300 K surface temperature over all three modes.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Iso</td>
<td>18.5</td>
<td>54.6</td>
<td>185</td>
</tr>
<tr>
<td>Iso–PT²</td>
<td>8.28</td>
<td>21.2</td>
<td>71.9</td>
</tr>
<tr>
<td>Iso–PT³</td>
<td>8.38</td>
<td>21.2</td>
<td>72.0</td>
</tr>
<tr>
<td>Iso–LPT_{BL}</td>
<td><strong>7.36</strong></td>
<td><strong>16.4</strong></td>
<td><strong>55.5</strong></td>
</tr>
<tr>
<td>Iso–LPT_{EU}</td>
<td>9.78</td>
<td>23.0</td>
<td>78.1</td>
</tr>
<tr>
<td>Iso–LPT_{T∞}</td>
<td>9.96</td>
<td>23.0</td>
<td>78.1</td>
</tr>
<tr>
<td>Poly</td>
<td>18.6</td>
<td>54.6</td>
<td>185</td>
</tr>
<tr>
<td>Poly–PT²</td>
<td>8.66</td>
<td>21.2</td>
<td>71.8</td>
</tr>
<tr>
<td>Poly–PT³</td>
<td>8.77</td>
<td>21.2</td>
<td>71.9</td>
</tr>
<tr>
<td>Poly–LPT_{BL}</td>
<td><strong>7.75</strong></td>
<td><strong>16.9</strong></td>
<td><strong>54.5</strong></td>
</tr>
<tr>
<td>Poly–LPT_{EU}</td>
<td>10.2</td>
<td>23.0</td>
<td>78.0</td>
</tr>
<tr>
<td>Poly–LPT_{T∞}</td>
<td>10.4</td>
<td>23.0</td>
<td>78.0</td>
</tr>
</tbody>
</table>
“NS”, “NS_\text{ss}”, the steady-state surrogates (“Iso” and “Poly”) and the corrections: “PT^2” and “LPT_{\text{BL}}”. The “LPT_{\text{BL}}” model for both surrogates had the lowest error of all of the methods considered. The results for the “Iso” model for oscillations of the first three modes are shown in Fig. 6.7, and for the “Poly” model in Fig. 6.8. The full GAFs are shown in figures (a), (c), and (e) for all three modes, while (b), (d), and (f) are the unsteady components of the GAFs.

The surrogate models with correction produce similar levels of error as the steady-state Navier-Stokes model with correction. It is interesting that the “Poly” model which performed poorly in the aerodynamic heating analysis, produces almost identical levels of error as the “Iso” model for the pressure. This again indicates that aerodynamic pressure is relatively insensitive to surface temperature, and thus the differences in the development of these two models has little impact on modeling the aerodynamic pressure.
Figure 6.7: Navier-Stokes and the corrected steady-state pointwise surrogate Generalized Aerodynamic Forces (GAFs) for the HSV panel: \( f = 140 \) Hz, \( M_2 = 6.86, T_2 = 300K, P_2 = 2966Pa, T_w = 300K, T_r = 0.085. \)
Figure 6.8: Navier-Stokes and the corrected polynomial surface temperature basis surrogate Generalized Aerodynamic Forces (GAFs) for the HSV panel: $f = 140\, Hz$, $M_2 = 6.86$, $T_2 = 300K$, $P_2 = 2966\, Pa$, $T_w = 300K$, $T_r = 0.085$. 
6.3 Fluid-Thermal-Structural Method of Solution

The partitioned F-T-S solution process originally developed by Culler and McMara [21] is summarized here. For a complete description, including verification and validation of the model and approach, refer to [21].

The framework of the aerothermoelastic model [21], shown in Fig. 6.9, is divided into aerothermal and aeroelastic components. The structural temperature distribution is passed from the aerothermal solution to the aeroelastic problem in path (1). Feedback of elastic deformation to the aerothermal problem is transferred in path (2). The partitioned time-marching approach [21] is illustrated in Fig. 6.10. Step (1) is a time marching solution of the aeroelastic problem to proceed from time \( t \) to time \( t + \Delta t_{AT} \). In step (2) the elastic deformation that occurred during step (1) is passed to the aerothermal model in a time-averaged sense [21]. Step (3) is a time marching solution of the aerothermal problem to proceed from time \( t \) to time \( t + \Delta t_{AT} \). Finally, in step (4) the structural temperature distribution is updated in the aeroelastic model. Note that different steps sizes are used for the aerothermal and aeroelastic time marching process.

The structural equation of motion is based on a 2-D von Kármán panel [21]:

\[
\frac{\partial^2}{\partial x^2} \left( D \frac{\partial^2 w}{\partial x^2} \right) - N_x \frac{\partial^2 w}{\partial x^2} + \left( \sum_i h_i \rho_i \right) \frac{\partial^2 w}{\partial t^2} + P_w - P_\infty + \frac{\partial^2 M_T}{\partial x^2} = 0
\] (6.1)

This equation of motion is discretized using Galerkin’s method to replace the spatial dependence with a summation of six assumed sine modes. Sine modes are
Figure 6.9: Components of the aerothermoelastic model of the HSV panel [21].

Figure 6.10: Aerothermoelastic solution using a partitioned approach to couple aerothermal and time-averaged aeroelastic solutions for the HSV panel model [21].
chosen because they satisfy the geometric and natural boundary conditions of this simply-supported, semi-infinite panel [21]. The resulting system of nonlinear, ordinary differential equations is integrated directly in the time domain using a fourth order Runge-Kutta method [21].

Transient heat transfer to the panel is computed using the a finite difference solution to the 2-D heat equation for the chord-wise and through-thickness directions [21]:

\[ \rho c \frac{\partial T}{\partial t} = k_x \frac{\partial^2 T}{\partial x^2} + k_z \frac{\partial^2 T}{\partial z^2} \]  

(6.2)

As shown in Fig. 6.11, the thermal model [21] for the thermal protection system and panel is assumed to consist of three layers: 1) a radiation shield (PM-2000 honeycomb sandwich), 2) thermal insulation (internal multi-screen insulation), and 3) panel structure (a high temperature grade titanium alloy). Boundary conditions for the thermal model consist of aerodynamic heating and thermal radiation at the upper surface of the radiation shield, and adiabatic conditions at the panel edges and bottom surface. The thermophysical properties and thickness of each layer [21] are listed in Table 6.5. Note that specific heat and thermal conductivity are temperature dependent. Data for these properties as a function of temperature are provided by Ref. [124] for the radiation shield and the thermal insulation, and by Ref. [125] for the plate structure. Finally, note that the mass of the thermal protection system is included in Eq. (6.1), but the stiffness is neglected.

The computation of the CFD-based generalized aerodynamic forces on the panel requires spatial integration of the surface pressure. This integration is carried out
Figure 6.11: Two-dimensional model of the thermal structure for the HSV panel [21].

Table 6.5: Properties of the thermal structure of the HSV panel at 300 K [21].

<table>
<thead>
<tr>
<th></th>
<th>$\rho$</th>
<th>$c$</th>
<th>$k$</th>
<th>$h_i$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(kg/m$^3$)</td>
<td>(J/kg/K)</td>
<td>(W/m/K)</td>
<td>(mm)</td>
</tr>
<tr>
<td>Radiation Shield</td>
<td>1010</td>
<td>465</td>
<td>0.250</td>
<td>2.0</td>
</tr>
<tr>
<td>Thermal Insulation</td>
<td>73.0</td>
<td>729</td>
<td>0.0258</td>
<td>10.0</td>
</tr>
<tr>
<td>Plate Structure</td>
<td>4540</td>
<td>463</td>
<td>6.89</td>
<td>5.0</td>
</tr>
</tbody>
</table>
numerically with a two point Gaussian quadrature with 500 surface elements since this yielded convergence of the aerothermoelastic flutter boundary within 0.12% at Mach 10.5 when compared to a 1000 surface element integration. Consistent with Culler and McNamara [21], only deformation induced pressures are included in the aerothermoelastic analysis. Thus, the non-zero, stream-wise pressure variations on the undeformed panel surface, which occur due to boundary layer displacement effects, are removed by subtracting the undeformed surface pressure at each time step of the aerothermoelastic analysis.

6.4 Fluid-Thermal-Structural Analysis

The F-T-S response of the panel is computed using Eqs. (6.1) and (6.2) with four different models for the aerothermodynamic loads. The first is the original model used by Culler and McNamara [21] ("C–M"): third-order piston theory for pressure and Eckert’s reference enthalpy for the aerodynamic heating. Additionally, the corrected isothermal model ("CIM"), linear heat transfer model ("Linear"), and the polynomial temperature basis model ("Poly") are also included. In terms of modeling the pressure, the most accurate model is implemented the surrogate models; the local piston theory with $T_{loc}$ evaluated at the approximate boundary layer edge ("LPT_{BL}"). Note, both the "CIM" and "Linear" models use the same pressure model in order to solely assess the impact of different aerodynamic heating models in the response. Note, the model names and associated sub-models
for the aerodynamic heating and pressure are listed in Table 6.6. Comparisons between the different modeling approaches are made for the flutter boundary, dynamically stable F-T-S response, and post-flutter F-T-S response. The panel configuration and operating conditions used for F-T-S analysis are listed in Table 6.7.

Table 6.6: Aerodynamic heating and pressure sub-models for the fluid-thermal-structural response of the HSV panel.

<table>
<thead>
<tr>
<th>Case</th>
<th>Aerodynamic Heating</th>
<th>Aerodynamic Pressure</th>
</tr>
</thead>
<tbody>
<tr>
<td>C–M, Ref. [21]</td>
<td>Eckert’s Ref. Enthalpy</td>
<td>PT³</td>
</tr>
<tr>
<td>CIM</td>
<td>CIM</td>
<td>Iso–LPT₁BL</td>
</tr>
<tr>
<td>Linear</td>
<td>Linear</td>
<td>Iso–LPT₁BL</td>
</tr>
<tr>
<td>Poly</td>
<td>Poly</td>
<td>Poly–LPT₁BL</td>
</tr>
</tbody>
</table>

Table 6.7: Parameters used to compute the fluid-thermal-structural response of the HSV panel.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Altitude (km)</td>
<td>30</td>
</tr>
<tr>
<td>Free stream Mach Number, ( M_1 )</td>
<td>7.5 – 10.5</td>
</tr>
<tr>
<td>Initial Panel Temperature, ( T_w ) (K)</td>
<td>300</td>
</tr>
</tbody>
</table>
6.5 Flutter Boundary

As the panel heats up due to aerodynamic heating, the thermal moment forces the panel to buckle into the flow. Due to the increasing temperature in the panel and the increasing aerodynamic pressure, the panel may reach a point of instability where the panel will snap out of the flow and proceed into self-excited oscillations. The location of this instability is known as the flutter boundary. The flight time to the onset of flutter for the panel for seven different constant Mach number trajectories is shown in Fig. 6.12. The differences in flight time compared to the original “C–M” model are listed in Table 6.8. Note, the “Poly” model failed to predict flutter and results from this model are presented at the end of this chapter.

As noted in Ref. [21], the higher the free stream Mach number, the shorter the onset time to panel flutter. It is interesting that the differences between the “C–M” and “CIM” aerothermodynamic modeling approaches decrease with free stream Mach number. At the highest Mach number considered ($M_1 = 10.5$), the difference between the two methods is almost 100%; while at the lowest Mach number considered ($M_1 = 7.5$), the difference is less than 1%. Furthermore, the analytical modeling approach yielded a conservative estimate for the flutter boundary for all of the Mach numbers considered relative to the “CIM” case.

The linear heat transfer case (“Linear”), also provided a conservative estimate of the flutter boundary compared to the “CIM” case for all Mach numbers; predicting the flutter boundary almost 20% earlier at $M_1 = 8.5$. The “Linear” case predicted
flutter boundaries between the “CIM” and “C–M” cases from $M_1 = 10.5$ down to $M_1 = 9.0$, where it becomes the most conservative model. Further insight into the sources of differences between the flutter boundaries is provided by a comparison of the dynamically stable fluid-thermal-structural panel response discussed next.

![Figure 6.12: Comparison of flight times to the onset of flutter predicted using the “CIM”, “Linear”, and analytical model used by Culler and McNamara [21] (“C–M”).](image)

Figure 6.12: Comparison of flight times to the onset of flutter predicted using the “CIM”, “Linear”, and analytical model used by Culler and McNamara [21] (“C–M”).
Table 6.8: Percent difference in the flight times to the onset of flutter predicted using the “CIM”, “Linear”, and analytical models used by Culler and McNamara [21] (“C–M”).

<table>
<thead>
<tr>
<th>Mach Number</th>
<th>CIM</th>
<th>Linear</th>
<th>CIM</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>C–M</td>
<td>C–M</td>
<td>Linear</td>
</tr>
<tr>
<td>$M_1 = 7.5$</td>
<td>0.12%</td>
<td>-9.57%</td>
<td>10.71%</td>
</tr>
<tr>
<td>$M_1 = 8.0$</td>
<td>3.83%</td>
<td>-10.85%</td>
<td>16.47%</td>
</tr>
<tr>
<td>$M_1 = 8.5$</td>
<td>8.65%</td>
<td>-9.27%</td>
<td>19.76%</td>
</tr>
<tr>
<td>$M_1 = 9.0$</td>
<td>18.76%</td>
<td>-0.19%</td>
<td>18.99%</td>
</tr>
<tr>
<td>$M_1 = 9.5$</td>
<td>37.49%</td>
<td>18.60%</td>
<td>15.92%</td>
</tr>
<tr>
<td>$M_1 = 10.0$</td>
<td>64.53%</td>
<td>47.29%</td>
<td>11.70%</td>
</tr>
<tr>
<td>$M_1 = 10.5$</td>
<td>96.61%</td>
<td>84.04%</td>
<td>6.83%</td>
</tr>
</tbody>
</table>
6.6 Dynamically Stable Response

Snap shots of the dynamically stable panel response are shown in Fig. 6.13 and 6.14. Differences between the “CIM”, “Linear”, and the “C–M” results near the flutter boundary of the “C–M” model are quantified in Tables 6.9 and 6.10 for the $M_1 = 7.5$ and 10.5 trajectories, using Eqs. (3.22) to (3.25). These two trajectories are chosen since they resulted in the least and greatest difference between the analytical “C–M” and CFD-based “CIM” aerothermodynamic models in terms of flight time to the onset of flutter. Five different quantities are considered for each trajectory: pressure differential, aerodynamic heat flux, panel displacement, mid-plate temperature, and average mid-plate temperature. The first two quantities represent the driving loads in the system, while the second three represent the panel response. Note, the average mid-plate temperature is shown as a function of time, with the symbols indicating when flutter occurred for each model.

For the $M_1 = 7.5$ trajectory, there are modest differences in the panel loads and response between the “CIM” and “C–M” cases. For example, the average differences between the loads for two models are under 25% and deformation and temperature rise are under 5%. The differences are much larger between the “Linear” and “C–M” cases, with average differences up to $\sim 35\%$ and maximum differences of $\sim 100\%$. For the $M_1 = 10.5$ case, the differences between the “CIM” and “C–M” models are larger in terms of panel displacement, similar in terms of pressure, and smaller in terms of heat flux and the mid-plate temperature. The difference
Figure 6.13: Pressure, heating, panel displacement, and thermal loads during dynamically stable fluid-thermal-structural response of the HSV panel at $M_1 = 7.5$. 

166
Figure 6.14: Pressure, heating, panel displacement, and thermal loads during dynamically stable fluid-thermal-structural response of the HSV panel at $M_1 = 10.5$. 

(a) Aerodynamic Pressure. 
(b) Aerodynamic Heat Flux. 
(c) Panel Displacement. 
(d) Mid-Plate Temperature. 
(e) Average Mid-Plate temperature.
Table 6.9: Percent difference between the “CIM” and “Linear” models from the “C–M” model for pressure, heating, panel displacement, and mid-plate temperature rise from 300 K for $M_1 = 7.5$ at 1500 sec.

<table>
<thead>
<tr>
<th></th>
<th>CIM</th>
<th>Linear</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Avg. Diff. (%)</td>
<td>Max Diff. (%)</td>
</tr>
<tr>
<td>Pressure</td>
<td>24.2</td>
<td>51.7</td>
</tr>
<tr>
<td>Heat Flux</td>
<td>16.7</td>
<td>23.6</td>
</tr>
<tr>
<td>Displacement</td>
<td>2.49</td>
<td>6.17</td>
</tr>
<tr>
<td>Mid-plate Temp.</td>
<td>4.30</td>
<td>5.34</td>
</tr>
</tbody>
</table>

Table 6.10: Percent difference the “CIM” and “Linear” models from the “C–M” model for pressure, heating, panel displacement, and mid-plate temperature rise from 300 K for $M_1 = 10.5$ at 40 sec.

<table>
<thead>
<tr>
<th></th>
<th>CIM</th>
<th>Linear</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Avg. Diff. (%)</td>
<td>Max Diff. (%)</td>
</tr>
<tr>
<td>Pressure</td>
<td>28.7</td>
<td>49.2</td>
</tr>
<tr>
<td>Heat Flux</td>
<td>9.47</td>
<td>12.5</td>
</tr>
<tr>
<td>Displacement</td>
<td>25.6</td>
<td>52.6</td>
</tr>
<tr>
<td>Mid-plate Temp.</td>
<td>0.19</td>
<td>0.23</td>
</tr>
</tbody>
</table>
in the mid-plate temperature is small, due to the short time duration. Also note, differences in panel displacement near the onset of flutter for the “C–M” case are much higher for $M_1 = 10.5$, max of 53%, versus the $M_1 = 7.5$ case, max of 6%. The differences between the “Linear” and “C–M” models are larger on average for all four quantities for $M_1 = 10.5$ compared to $M_1 = 7.5$. Note, consistently for both Mach numbers, the “Linear” model predicts the highest average mid-plate temperatures, followed by the “CIM” and then the “C–M” model. It is interesting that at $M_1 = 7.5$ flutter occurs at approximately the same time for both the “CIM” and “C–M” models even though the “CIM” case has an average mid-plate temperature $\sim$8% higher at the point of flutter, as shown in Fig. 6.13(e). At $M_1 = 10.5$ the two models differ in flutter time by almost 100%, and “CIM” case has an approximately 450% greater temperature rise prior to flutter than the “C–M” model, as shown in Fig. 6.14(e). These differences illustrate the complexity and coupled nature of fluid-thermal-structural responses, and the need to predict the aerothermodynamic loads accurately.

While these comparisons indicate some increased differences between the CFD-based models and the analytical model at the different Mach numbers, they do not fully explain the large discrepancy in onset time to flutter at $M_1 = 10.5$. However, a comparison of the relative magnitude of each quantity at Mach 10.5 and 7.5, shown in Fig. 6.15, provides additional insight. Note that the $M_1 = 7.5$ trajectory produces significantly higher panel displacements, mid-plate temperatures, and pressure differentials, but smaller surface heat flux. The lower heat flux is due to operation
at a lower free stream Mach number at Mach 7.5 compared to \( M_1 = 10.5 \) and elevated surface temperatures. Despite the lower heating rate, the significantly delayed flutter of the panel at \( M_1 = 7.5 \) compared to \( M_1 = 10.5 \) results in the panel ultimately reaching a higher temperature.

It is counter-intuitive that the \( M_1 = 7.5 \) case withstands higher pressures prior to loss of dynamic stability compared to the \( M_1 = 10.5 \) case, while at a significantly elevated temperature. This indicates that the \( M_1 = 7.5 \) panel case experiences significant stiffening during post-buckling as a result of the in-plane thermal force, and that the rate of stiffening occurs at a faster rate than the accumulation of pressure loading and reduction in Young’s modulus due to temperature rise. This ultimately enables the panel to sustain higher pressure loading prior to initiating limit cycle oscillations. Such an explanation is supported by previous research [126, 127], which has shown that fundamental frequencies of simply-supported beams and plates increase with increasing in-plane thermal load during post-buckled displacements. Since the panel is stiffer at \( M_1 = 7.5 \), compared to \( M_1 = 10.5 \), it is less sensitive to differences in pressure between the CFD-based “CIM” and “C–M” models, reducing the difference in the onset time to flutter.

### 6.7 Nonlinear Post-Flutter Response

A comparison of the nonlinear panel flutter response, represented using the panel displacement is shown in Fig. 6.16 and 6.17 for the \( M_1 = 7.5 \) and \( M_1 = 10.5 \) trajec-
Figure 6.15: Pressure, heating, panel displacement, and thermal loads near the onset time of flutter for both the $M_1 = 7.5$ (1500 sec.) and $M_1 = 10.5$ (40 sec) trajectories of the HSV panel.
tories, respectively. The displacement envelopes, representing the maximum and minimum panel displacement, are shown in Figs. 6.16(a) and 6.17(a). Following [21], four distinct regions of response are evident, namely: 1) initially flat and stable, followed by 2) dynamically stable deformation, before 3) the onset of flutter, and a transition to 4) limit cycle oscillations. In Fig. 6.17(a), the panel response computed using the both the CFD-based models ("CIM" and "Linear") remain in the buckled but dynamically stable configuration much longer than predicted in Ref. [21]. Note, at the point that oscillations begin, the "Linear" model starts from a larger deformation, which impacts the post flutter response and limit cycle oscillations.

The post flutter oscillations for each Mach number are shown in Figs. 6.16(b) and 6.17(b), and the instantaneous panel deformation during limit cycle oscillation is illustrated in Figs. 6.16(c) and 6.17(c). Note that the time scale in Figs. 6.16(b) and 6.17(b) are reduced compared to Figs. 6.16(a) and 6.17(a) in order to clearly discern the oscillations. In Figs. 6.16(c) the "CIM", "Linear", and "C–M" models oscillate primarily in the the fifth mode, thus deformations in Figs. 6.16(b) and (c) are presented at the 50% chord location. While at $M_1 = 10.5$, the "CIM" and "C–M" models oscillate primarily in the second mode, and the "Linear" oscillates primarily in the third mode, thus the deformations in Figs. 6.17(b) and (c) are presented at the 75% chord location. For the $M_1 = 10.5$ response, comparing the limit cycle oscillations, Fig. 6.17(b), to the instantaneous deformation during limit cycle oscillations, Fig. 6.17(c), reveals that the "CIM", "C–M" models predict roughly the
Figure 6.16: Nonlinear fluid-thermal-structural panel flutter response at $M_1 = 7.5$. 

(a) Displacement envelope ($M_1 = 7.5$).

(b) Post flutter oscillations ($M_1 = 7.5$).

(c) Instantaneous deformation ($M_1 = 7.5$).
(a) Displacement envelope ($M_1 = 10.5$).

(b) Post flutter oscillations ($M_1 = 10.5$).

(c) Instantaneous deformation ($M_1 = 10.5$).

Figure 6.17: Nonlinear fluid-thermal-structural panel flutter response at $M_1 = 10.5$. 
same frequency of oscillation 140 Hz, and the “Linear” model, even though it has a different dominant mode and different amplitude of oscillation, predicts a similar frequency of 135 Hz. For the $M_1 = 7.5$ response, Figs. 6.16(b,c), all of the models predict the frequency of oscillation to be approximately 625 Hz. Note that the frequency and slope of the $M_1 = 7.5$ post-flutter response exceed the well-known limits of classical piston theory [21, 22].

Finally, the “Poly” model response is compared to the “C–M” in Fig. 6.18. The displacement envelopes for $M_1 = 7.5$ and $M_1 = 10.5$ are shown in Fig. 6.18(a) and (b), respectively. The average mid-plate temperatures for both Mach numbers are shown in (c) and (d). The “Poly” model predicts an initial snap through and then proceeds to buckle out of the flow for both Mach numbers. These results are illustrative of the fluid-thermal-structural response predicted from a model with large errors, as detailed for the aerodynamic heating in Section 6.1.
(a) Displacement envelope \((M_1 = 7.5)\).

(b) Displacement envelope \((M_1 = 10.5)\).

(c) Average Mid-Plate temperature \((M_1 = 7.5)\).

(d) Average Mid-Plate temperature \((M_1 = 10.5)\).

Figure 6.18: Fluid-thermal-structural HSV panel response for the polynomial temperature basis surrogate.
Chapter 7

Fluid-Thermal-Structural Analysis: Shock Impingement Panel

In this study, the fluid-thermal-structural analysis of a thermally and structurally compliant panel with oscillating shock impingement is considered. First, two-dimensional aerodynamic heating and unsteady aerodynamic pressure surrogates are constructed. Next, the fluid-thermal-structural method of solution is introduced. Then, fluid-thermal and fluid-thermal-structural results are presented. Lastly, a similar three-dimensional configuration is considered, and aerodynamic heating and unsteady pressure surrogates are again constructed.

7.1 Aerodynamic Heat Flux Modeling

Turbulent aerodynamic heating surrogates are considered for the 2-D deformable panel subject to shock impingement described in Section 2.3. For this analysis, the free stream flow properties are held constant at Mach 3.0 with densities and temperatures corresponding to an altitude of 24 km. The parameters and corresponding bounds for this configuration are listed in Table 2.5. Aerodynamic heat
flux surrogate construction and verification is discussed next, followed by evaluation of the models relative to 100 full-order CFD test cases with variable surface temperatures, surface deformations, and shock impingement locations. For this analysis, the Menter $k - \omega$ SST turbulence model is used in all CFD training and evaluation cases.

7.1.1 Construction and Verification of CFD Surrogates

The aerodynamic heating models considered for this analysis are: the “Linear” model, pointwise “Iso” model, “Poly” model, and the “CIM”.

Linearized Model

The model with linearized heat flux with respect to surface temperature approximation (“Linear”) neglects deformation as well as variable shock impingement location. Since free stream parameters are not varied in this analysis, surrogate models for the heat transfer coefficient and adiabatic wall temperature are unnecessary. Thus, the heat transfer coefficient is computed directly from Eq. (4.4) using 2 CFD training solutions: (1) surface temperature set to free stream and (2) surface temperature set to adiabatic. The aerodynamic heating is then computed using Eq. (4.5). Note, the shock generator angle is held constant at $10^\circ$ in both of the training solutions.
**Pointwise Model**

The pointwise surrogate ("Iso") is constructed from 1500 CFD training solutions in order to account for both deformation and shock impingement location. The input parameters are: isothermal surface temperature; surface deformation modal amplitudes, $a_1^*-a_6^*$; and shock generator angle. Verification of the model is carried out using 25 full-order isothermal CFD solutions from the same parameter space. The resulting errors are a mean of 4.68% and a max of 128%. The maximum dimensional error is $0.98 \text{W/cm}^2$ relative to a test case with a peak heat flux of $4.62 \text{W/cm}^2$. Note, that large maximum errors occur due to sharp gradients in the heat flux at the shock impingement location. Thus, the normalized mean error is the considered more representative of model quality, since it is not sensitive to small deviations in the shock impingement location. Note that a comparison of the peak heat flux prediction between the surrogate and CFD, revealed only a 1.3% difference for the test case producing the largest dimensional error.

**Parameterization of the Surface Temperature using a Polynomial Basis**

The input parameters for "Poly" surrogate are: polynomial coefficients, $b_0-b_6$; surface deformation modal amplitudes, $a_1^*-a_6^*$; and shock generator angle. The bounds on $b_0$ are set in accordance with the original bounds of the surface temperature listed in Table 2.5. The bounds on $b_1-b_6$ are set similarly to the flat panel analysis, $\pm 1000$. The model is then generated from 1500 CFD training solutions. Verifica-
tion of the model is carried out using 25 full-order CFD solutions from the same parameter space. The resulting errors are a mean of 15.3%, max of 292%, and a maximum dimensional error of 2.26 $W/cm^2$ relative to a test case with a peak heat flux of 4.25 $W/cm^2$. Similar to the “Iso” surrogate, small differences in the predicted location of the shock impingement resulted in large maximum errors. Comparing peak heat flux predictions, for the test case with the largest dimensional error, resulted in a 6.2% difference between the “Poly” surrogate and CFD.

**Corrected Pointwise Isothermal Model**

The corrected pointwise isothermal model (“CIM”) is also examined using the Menter coefficients from Table 4.3. As before, the “Iso” surrogate is used as a baseline predictor for the heat flux. One complication with including deformation and shock impingement is that the velocity, density, and temperature at the edge of the boundary layer will vary chordwise along the panel. These quantities are readily approximated using pointwise surrogates developed from isothermal training cases. However an additional prediction is required in order to determine the location of boundary layer edge. This is approximated [123] using Eq. (5.17). Verification of the pointwise surrogates for predicting the density, temperature, and velocity at the boundary layer edge is carried out through comparison with 25 full-order isothermal test cases with varying surface deformation, shock impingement location. This yielded mean errors of 2.65% for density, 1.10% for temperature, and 0.38% for velocity.
7.1.2 Comparison of Aerodynamic Heating Models

In order to compare all of these different models, 100 CFD evaluation cases are computed for turbulent flow, varying surface temperature, surface deformation, and shock generator angle. The test cases are chosen through Latin hypercube sampling of the parameter space. The mean, maximum percent, maximum dimensional error are listed in Table 7.1. The “CIM” surrogate yields the lowest error by all metrics. As expected, the “Iso” surrogate yields higher errors since it ignores spatial gradients in the surface temperature. Note that the relatively high maximum percent error for the “CIM” surrogate is driven by maximum percent errors in the “Iso” model that result from small deviations in the shock impingement location. Note that the “Poly” model has the largest mean and maximum percent error of those considered.

Table 7.1: Error in the aerodynamic heat flux models for the 2-D deformable panel subject to shock impingement compared to CFD.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (W/cm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>31.3</td>
<td>196</td>
<td>3.14</td>
</tr>
<tr>
<td>Iso</td>
<td>12.4</td>
<td>189</td>
<td>0.77</td>
</tr>
<tr>
<td>CIM</td>
<td><strong>4.24</strong></td>
<td><strong>118</strong></td>
<td><strong>0.69</strong></td>
</tr>
<tr>
<td>Poly</td>
<td>76.0</td>
<td>871</td>
<td>3.09</td>
</tr>
</tbody>
</table>

A comparison of the different models are shown in Fig. 7.1, where the corresponding temperature profile and surface deformation are also included. Note
that these conditions correspond with those that resulted in the maximum dimensional error for the “CIM”. For this particular test case, both the “CIM” and “Iso” models have maximum errors below 0.75 W/cm², which occur at the shock impingement location. The “Linear” model does not capture the variation in heat flux due to deformation or variable shock impingement location, leading to high errors. While a sixth order polynomial captures the temperature profile reasonably well, it is clear that the “Poly” model is incapable of reliably reproducing any of the complex features in the heat flux prediction.
Figure 7.1: Comparison of several turbulent aerodynamic heating predictions to the Benchmark (CFD) for the 2-D deformable panel subject to shock impingement: $M_\infty = 3.0$, Alt. = 24 km, Shock Generator Angle = 9.90°.
7.2 Unsteady Aerodynamic Pressure Modeling

The unsteady aerodynamic pressure resulting from a shock impinging on an oscillating surface is considered next. For this simulation two configurations are considered; the effect of a stationary shock and the effect of an oscillating shock. The free stream properties are held constant at $M_\infty = 3.0$ and 24 km altitude, and the surface temperature is set to two times the free stream, 433 K.

7.2.1 Stationary Shock

For the stationary shock, three cases are included: the first, second, and third clamped-clamped panel modes oscillating independently at their respective natural frequencies, listed in Table 2.5. The amplitudes are: $a_1^* = 5.0$, $a_2^* = 1.0$, and $a_3^* = 0.2$. The shock generator is held constant at $10^\circ$. GAFs are computed for the unsteady Euler (“EU”), second- and third-order piston theory (“PT$^2$” and “PT$^3$”), steady-state Euler (“EU$\text{SS}$”), and corrections for the steady-state Euler using piston theory and local piston theory.

The results are listed in Table 7.2 and shown in Figs. 7.2 and 7.3. The classical piston theory models do not model the impact of shock-impingement and have approximately 100% error for the full GAFs, Fig. 7.2 (a,c,e). However the unsteady components shown in (b,d,f), compare favorably with the unsteady component of Euler. The local piston theory models all have mean errors under 0.05% and maximum errors under 0.15%, with the “LPT$\text{EU}$” model providing the lowest er-
rors. These models, shown in Fig. 7.3, have negligible error in the prediction of the unsteady component of the GAFs (b,d,f).

Table 7.2: Error in GAFs for a stationary shock impinging on a 2-D panel, comparing analytical and Euler-based methods for oscillations of the first three panel modes.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>PT²</td>
<td>99.7</td>
<td>100</td>
<td>195</td>
</tr>
<tr>
<td>PT³</td>
<td>99.7</td>
<td>100</td>
<td>195</td>
</tr>
<tr>
<td>EU&lt;sub&gt;SS&lt;/sub&gt;</td>
<td>0.624</td>
<td>1.75</td>
<td>2.50</td>
</tr>
<tr>
<td>EU&lt;sub&gt;SS&lt;/sub&gt;–PT²</td>
<td>0.136</td>
<td>0.410</td>
<td>0.585</td>
</tr>
<tr>
<td>EU&lt;sub&gt;SS&lt;/sub&gt;–PT³</td>
<td>0.136</td>
<td>0.410</td>
<td>0.585</td>
</tr>
<tr>
<td>EU&lt;sub&gt;SS&lt;/sub&gt;–LPT&lt;sub&gt;EU&lt;/sub&gt;</td>
<td><strong>0.034</strong></td>
<td><strong>0.075</strong></td>
<td><strong>0.138</strong></td>
</tr>
<tr>
<td>EU&lt;sub&gt;SS&lt;/sub&gt;–LPT&lt;sub&gt;T∞&lt;/sub&gt;</td>
<td>0.049</td>
<td>0.131</td>
<td>0.254</td>
</tr>
</tbody>
</table>

Next, the same three cases are modeled using Navier-Stokes aerodynamics. GAFs are computed for the unsteady Navier-Stokes ("NS"), steady-state Navier-Stokes ("NS<sub>SS</sub>"), Euler, classical piston theory, and the unsteady corrections for the steady-state Navier-Stokes model. The results are listed in Table 7.3 and shown in Figs. 7.4 and 7.5. In Fig. 7.4 Navier-Stokes is compared with Euler and classical piston theory, however due to the large error of piston theory these models do not even appear in Figs. 7.4(a,c,e). The local piston theory corrections are shown in Fig. 7.5. Similar to the inviscid results, the local piston theory corrections to the steady state model yielded the best approximation of the fully unsteady model. Errors are slightly higher, 0.45% for the mean and 2.3% for the maximum.
Figure 7.2: Euler and piston theory Generalized Aerodynamic Forces (GAFs) for an oscillating panel with a stationary shock. $f_1 = 61.01 \text{ Hz}$, $f_2 = 168.2 \text{ Hz}$, $f_3 = 329.7 \text{ Hz}$, $M_\infty = 3.0$, $Alt. = 24 \text{ km}$. 

(a) GAFs Mode 1. $\alpha_1* = 5.0$
(b) Unsteady Component of GAFs, Mode 1.

(c) GAFs Mode 2. $\alpha_2* = 1.0$
(d) Unsteady Component of GAFs, Mode 2.

(e) GAFs Mode 3. $\alpha_3* = 0.2$
(f) Unsteady Component of GAFs, Mode 3.
Figure 7.3: Euler and local piston theory Generalized Aerodynamic Forces (GAFs) for an oscillating panel with a stationary shock. $f_1 = 61.01 \ Hz$, $f_2 = 168.2 \ Hz$, $f_3 = 329.7 \ Hz$, $M_\infty = 3.0$, Alt. = 24 km.

(a) GAFs Mode 1. $a_1^* = 5.0$

(b) Unsteady Component of GAFs, Mode 1.

(c) GAFs Mode 2. $a_2^* = 1.0$

(d) Unsteady Component of GAFs, Mode 2.

(e) GAFs Mode 3. $a_3^* = 0.2$

(f) Unsteady Component of GAFs, Mode 3.
Figure 7.4: Navier-Stokes, Euler, and piston theory Generalized Aerodynamic Forces (GAFs) for an oscillating panel with a stationary shock. $f_1 = 61.01 \text{ Hz}$, $f_2 = 168.2 \text{ Hz}$, $f_3 = 329.7 \text{ Hz}$, $M_\infty = 3.0$, Alt. = 24 km.
(a) GAFs Mode 1. $a_1^* = 5.0$

(b) Unsteady Component of GAFs, Mode 1.

(c) GAFs Mode 2. $a_2^* = 1.0$

(d) Unsteady Component of GAFs, Mode 2.

(e) GAFs Mode 3. $a_3^* = 0.2$

(f) Unsteady Component of GAFs, Mode 3.

Figure 7.5: Navier-Stokes and local piston theory Generalized Aerodynamic Forces (GAFs) for an oscillating panel with a stationary shock. $f_1 = 61.01$ Hz, $f_2 = 168.2$ Hz, $f_3 = 329.7$ Hz, $M_\infty = 3.0$, Alt. = 24 km.
Table 7.3: Error in GAFs for a stationary shock impinging on a 2-D panel, comparing Navier-Stokes, analytical, and Euler-based methods for oscillations of the first three panel modes.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EU</td>
<td>510</td>
<td>1426</td>
<td>150</td>
</tr>
<tr>
<td>PT²</td>
<td>100</td>
<td>116</td>
<td>346</td>
</tr>
<tr>
<td>PT³</td>
<td>100</td>
<td>116</td>
<td>346</td>
</tr>
<tr>
<td>NSₜₜ</td>
<td>2.01</td>
<td>7.91</td>
<td>3.62</td>
</tr>
<tr>
<td>NSₜₜ–PT²</td>
<td>1.06</td>
<td>4.22</td>
<td>2.41</td>
</tr>
<tr>
<td>NSₜₜ–PT³</td>
<td>1.06</td>
<td>4.22</td>
<td>2.41</td>
</tr>
<tr>
<td>NSₜₜ–LPTₜBL</td>
<td>0.455</td>
<td>2.31</td>
<td>1.16</td>
</tr>
<tr>
<td>NSₜₜ–LPTₜEU</td>
<td>0.445</td>
<td><strong>2.28</strong></td>
<td>1.06</td>
</tr>
<tr>
<td>NSₜₜ–LPTₜ∞</td>
<td><strong>0.443</strong></td>
<td>2.30</td>
<td><strong>0.970</strong></td>
</tr>
</tbody>
</table>

Next, the surrogate models are introduced for this problem. The steady-state “Iso” and steady-state “Poly” models with and without unsteady correction are considered. The parameter space, 1500 CFD training snap shots, and development of both of these models for aerodynamic heating is discussed in Section 7.1. The steady-state aerodynamic pressure surrogates are developed from the same snap shots and input parameters as for the aerodynamic heating surrogates. These models along with the unsteady corrections are compared with the unsteady Navier-Stokes solution in Table 7.4 and in Figs. 7.6 and 7.7.

The errors for both the “Iso” and “Poly” models are relatively high, mean of 5% for the “Iso” and 16.6% for the “Poly”. The large error for the “Iso” methods is primarily due to oscillations of the third mode, Fig. 7.6 (e,f). The Navier-Stokes GAFs
are small and the steady-state “Iso” model prediction is in error by over 1 N from the steady-state Navier-Stokes model, leading to uncharacteristically high percent error. The overall dimensional maximum error occurred for the second modal analysis. However, the “Iso” corrected models all have under 0.5% mean and 2% maximum errors when considering only the first two mode results. The “Poly” model has large errors for both the second and third modal analyses, Figs. 7.7(c–f). Similar to the “Iso” model the high mean and maximum percent errors came from the third mode analysis, while the overall dimensional maximum error occurred for the mode two analysis. Note, unlike the “Iso” corrected models, the “Poly” corrected models still yield high error when considering only the first and second modal analyses, over 8% for the mean and over 26% for the maximum.
Table 7.4: Error in GAFs for a stationary shock impinging on a 2-D panel, comparing Navier-Stokes, and surrogate-based methods for oscillations of the first three panel modes.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Iso</td>
<td><strong>4.70</strong></td>
<td>20.6</td>
<td>2.81</td>
</tr>
<tr>
<td>Iso–PT²</td>
<td>4.74</td>
<td><strong>16.8</strong></td>
<td>1.56</td>
</tr>
<tr>
<td>Iso–PT³</td>
<td>4.74</td>
<td><strong>16.8</strong></td>
<td>1.56</td>
</tr>
<tr>
<td>Iso–LPT_{BL}</td>
<td>4.88</td>
<td>17.8</td>
<td>1.84</td>
</tr>
<tr>
<td>Iso–LPT_{EU}</td>
<td>4.89</td>
<td>18.0</td>
<td>1.84</td>
</tr>
<tr>
<td>Iso–LPT_{T∞}</td>
<td>4.91</td>
<td>18.2</td>
<td>1.91</td>
</tr>
<tr>
<td>Poly</td>
<td><strong>16.1</strong></td>
<td>39.3</td>
<td>30.5</td>
</tr>
<tr>
<td>Poly–PT²</td>
<td>16.4</td>
<td><strong>34.1</strong></td>
<td>30.4</td>
</tr>
<tr>
<td>Poly–PT³</td>
<td>16.4</td>
<td><strong>34.1</strong></td>
<td>30.4</td>
</tr>
<tr>
<td>Poly–LPT_{BL}</td>
<td>16.5</td>
<td>36.3</td>
<td>30.4</td>
</tr>
<tr>
<td>Poly–LPT_{EU}</td>
<td>16.5</td>
<td>36.4</td>
<td>30.4</td>
</tr>
<tr>
<td>Poly–LPT_{T∞}</td>
<td>16.5</td>
<td>36.7</td>
<td>30.4</td>
</tr>
</tbody>
</table>
Figure 7.6: Navier-Stokes and pointwise surrogate-based Generalized Aerodynamic Forces (GAFs) for an oscillating panel with a stationary shock. $f_1 = 61.01 \text{ Hz}$, $f_2 = 168.2 \text{ Hz}$, $f_3 = 329.7 \text{ Hz}$, $M_{\infty} = 3.0$, Alt. = 24 km.
Figure 7.7: Navier-Stokes and polynomial surface temperature surrogate-based Generalized Aerodynamic Forces (GAFs) for an oscillating panel with a stationary shock. $f_1 = 61.01 \ Hz$, $f_2 = 168.2 \ Hz$, $f_3 = 329.7 \ Hz$, $M_\infty = 3.0$, Alt. = 24 km.
7.2.2 Oscillating Shock

For the oscillating shock study only one case is considered. The panel is set to oscillate ±5 thicknesses at 50 Hz, and the shock-generator is set to oscillate at ±2° about a 10° nominal angle at 10 Hz. GAFs are computed for the unsteady Navier-Stokes ("NS"), Euler ("EU"), second- and third-order piston theory ("PT²" and "PT³"), steady-state Navier-Stokes ("NS SS"), and corrections for the steady-state Navier-Stokes using piston theory and local piston theory. The errors relative to the unsteady Navier-Stokes solution are listed in Table 7.5.

Table 7.5: Error in GAFs for an oscillating shock impinging on a vibrating 2-D panel, comparing Navier-Stokes, analytical, and Euler based methods for oscillations of the first panel mode.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EU</td>
<td>47.0</td>
<td>52.8</td>
<td>150</td>
</tr>
<tr>
<td>PT²</td>
<td>98.2</td>
<td>122</td>
<td>346</td>
</tr>
<tr>
<td>PT³</td>
<td>98.2</td>
<td>122</td>
<td>346</td>
</tr>
<tr>
<td>NS SS</td>
<td>0.535</td>
<td>1.27</td>
<td>3.62</td>
</tr>
<tr>
<td>NS SS–PT²</td>
<td>0.315</td>
<td>0.769</td>
<td>2.19</td>
</tr>
<tr>
<td>NS SS–PT³</td>
<td>0.315</td>
<td>0.769</td>
<td>2.18</td>
</tr>
<tr>
<td>NS SS–LPT_{BL}</td>
<td>0.287</td>
<td>0.564</td>
<td>1.60</td>
</tr>
<tr>
<td>NS SS–LPT_{EU}</td>
<td>0.293</td>
<td>0.590</td>
<td>1.68</td>
</tr>
<tr>
<td>NS SS–LPT_{T∞}</td>
<td>0.303</td>
<td>0.631</td>
<td>1.80</td>
</tr>
</tbody>
</table>

For visual comparison of the GAFs three cases are presented in Fig. 7.8, where (a), (c), and (e) are the full GAFs for the three cases and (b), (d), and (f) are the
unsteady components. The three cases are: first a comparison of Navier-Stokes, Euler, and piston theory methods, including: “NS”, “NS_{SS}”, “EU”, “EU_{SS}”, “PT^{2}”, and “PT^{3}”, shown in Figs. 7.8 (a) and (b). The second case is a comparison of Euler based methods only, Figs. 7.8 (c) and (d). Included in this analysis are the “EU”, “EU_{SS}”, and unsteady corrections to the steady-state Euler using second-order piston theory (“EU_{SS}-PT^{2}”), and local piston theory using steady-state Euler surface temperatures as the local temperature (“EU_{SS}-LPT_{EU}”). The third case only considers Navier-Stokes based models, shown in Figs. 7.8 (e) and (f). Included in this analysis are: “NS”, “NS_{SS}”, and unsteady corrections for the steady-state Navier-Stokes using second-order piston theory (“NS_{SS}-PT^{2}”), and local piston theory using steady-state Navier-Stokes local temperatures at the approximate boundary layer edge (“NS_{SS}-LPT_{BL}”).

For all of these cases, the unsteady components of the GAFs are approximately two-orders of magnitude smaller than the overall GAFs. Thus, the steady-state models have small error relative to their unsteady counterparts, less than 2% maximum error. However, there is a clear difference in the GAFs between analytical, Euler, and Navier-Stokes modeling of this problem, shown in Fig. 7.8 (a), where mean errors are approximately 50% or more. Clearly, the dominant feature of this problem is not the unsteady motion of the panel surface, but the inclusion of shock-impingement. The analytical methods do not capture the shock-impingement, while the Euler based methods do not capture it accurately relative to Navier-Stokes. However, the piston theory based correction represents a
Figure 7.8: Navier-Stokes, Euler, and approximate model Generalized Aerodynamic Forces (GAFs) for an oscillating shock impinging on a vibrating 2-D panel over 5 panel cycles. Shock generator: $\pm 2^\circ$ about a nominal $10^\circ$ angle at 10 Hz. Panel: $a'_f = 5$, $f = 50$ Hz. $M_\infty = 3.0$, Alt. = 24 km, $T_w = 433 K$. 

197
reasonably accurate and efficient method for incorporating unsteady pressure in a steady-state Navier-Stokes prediction. Note, the piston theory based methods predict sinusoidal GAFs with equal amplitudes for all time for the unsteady component, Fig. 7.8 (b), while the local piston theory methods have variable amplitude sinusoidal GAFs due to the variable local pressure, Figs. 7.8 (d) and (f), and are therefore slightly more accurate for this problem.

Finally, surrogate models are considered for the oscillating shock analysis. These models along with the unsteady corrections are compared with the unsteady Navier-Stokes solution in Table 7.6. The steady-state “Iso” model has similar errors as the steady-state Navier-Stokes model, approximately 1.4% maximum error; while the “Poly” model has errors as high as 8%. Additionally, when including the unsteady corrections, the “Iso”-based models have approximately 1% error; while the “Poly”-based models have errors around 8%.

For visual comparison, the surrogate-based GAFs are presented in Fig. 7.9, where (a) and (c) are the full GAFs, and (b) and (d) are the unsteady components. The “Iso”-based models are compared to the unsteady Navier-Stokes solution in Figs. 7.9 (a) and (b), and the “Poly”-based models are compared in Figs. 7.9 (c) and (d). The unsteady corrections for the surrogates are the second-order piston theory correction (“PT$^2$”), and local piston theory using local temperatures computed using the surrogates at the approximate boundary layer edge (“LPT$_{BL}$”). The “Iso”-based models, Figs. 7.9 (a) and (b), compare well with Navier-Stokes, and resemble the steady-state Navier-Stokes models with unsteady correction. The “Poly”-based
models, Figs. 7.9 (c) and (d), do show the overall correct trend due to the moving shock, albeit with larger errors.

Table 7.6: Error in GAFs for an oscillating shock impinging on a vibrating 2-D panel, comparing Navier-Stokes and surrogate-based methods for oscillations of the first panel mode.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Iso</td>
<td>0.542</td>
<td>1.36</td>
<td>3.86</td>
</tr>
<tr>
<td>Iso–PT²</td>
<td><strong>0.447</strong></td>
<td>1.01</td>
<td>2.87</td>
</tr>
<tr>
<td>Iso–PT³</td>
<td><strong>0.447</strong></td>
<td>1.01</td>
<td>2.87</td>
</tr>
<tr>
<td>Iso–LPT₁ encountered</td>
<td>0.454</td>
<td><strong>0.964</strong></td>
<td><strong>2.74</strong></td>
</tr>
<tr>
<td>Iso–LPT₁ EU</td>
<td>0.455</td>
<td>0.966</td>
<td>2.75</td>
</tr>
<tr>
<td>Iso–LPT₁ T∞</td>
<td>0.463</td>
<td>1.00</td>
<td>2.84</td>
</tr>
<tr>
<td>Poly</td>
<td>3.66</td>
<td>7.94</td>
<td>22.6</td>
</tr>
<tr>
<td>Poly–PT²</td>
<td>3.64</td>
<td>7.83</td>
<td>22.3</td>
</tr>
<tr>
<td>Poly–PT³</td>
<td>3.64</td>
<td>7.83</td>
<td>22.3</td>
</tr>
<tr>
<td>Poly–LPT₁</td>
<td><strong>3.63</strong></td>
<td>7.73</td>
<td>22.0</td>
</tr>
<tr>
<td>Poly–LPT₁ EU</td>
<td><strong>3.63</strong></td>
<td>7.73</td>
<td>22.0</td>
</tr>
<tr>
<td>Poly–LPT₁ T∞</td>
<td><strong>3.63</strong></td>
<td><strong>7.71</strong></td>
<td><strong>21.9</strong></td>
</tr>
</tbody>
</table>
Figure 7.9: Navier-Stokes and surrogate-based Generalized Aerodynamic Forces (GAFs) for an oscillating shock impinging on a vibrating 2-D panel over 5 panel cycles. Shock generator: $\pm 2^\circ$ about a nominal $10^\circ$ angle at $10 \text{ Hz}$. Panel: $a_1 = 5$, $f = 50 \text{ Hz}$. $M_\infty = 3.0$, $\text{Alt.} = 24 \text{ km}$, $T_w = 433K$. 

(a) Generalized Aerodynamic Forces.
(b) Unsteady Component of GAFs.
(c) Generalized Aerodynamic Forces.
(d) Unsteady Component of GAFs.
7.3 Fluid-Thermal-Structural Method of Solution

For the fluid-thermal-structural response of this panel, the thermal and structural models and method of solution were originally developed in a combined effort with Brent Miller in Ref. [128], and are briefly summarized here.

A schematic of the heat transfer model and structural model for the panel are shown in Figs. 7.10(a) and 7.10(b), respectively. The panel is assumed to be thermally insulated on the two ends as well as the side opposite of the flow. The panel is clamped on both ends so that the transverse displacements and their rotations are zero. The panel properties, shown in Table 7.7, are based on steel 4130 [125]. The numerical properties of the models are displayed in Table 7.8. The thermal solution is updated every 10 structural time steps. The number of mode shapes and size of the time steps were chosen based on a convergence study comparing the solution to a case using 40 modes and time steps \( \Delta t_S = \Delta t_T = 10 \mu s \). Differences in displacement and temperature rise between the two cases were negligible. The integration of the models and time marching scheme are described in Ref. [128].

The thermal model is constructed using a two dimensional finite element to solve the 2-D heat conduction equation, given as:

\[
[C] \{\dot{T}\} + [K] \{T\} = \{Q\}
\]  

(7.1)

where \([C]\), \([K]\), and \(\{Q\}\) are the thermal capacitance matrix, thermal conductivity matrix, and nodal heat load vector, respectively. The transient problem is then
Figure 7.10: Structural and thermal model configurations for the 2-D shock impingement panel.

Table 7.7: Heat conduction properties of the 2-D shock impingement panel at 220K.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \rho ) ( (kg/m^3) )</td>
<td>( 7.85 \times 10^3 )</td>
</tr>
<tr>
<td>( c ) ( (J/kg\cdot K) )</td>
<td>( 4.73 \times 10^2 )</td>
</tr>
<tr>
<td>( \kappa ) ( (J/m^2\cdot K) )</td>
<td>( 42.2 )</td>
</tr>
<tr>
<td>( \alpha ) ( (\mu m/mK) )</td>
<td>( 8.504 )</td>
</tr>
<tr>
<td>( E ) ( (GPa) )</td>
<td>( 203.8 )</td>
</tr>
<tr>
<td>( L ) ( (cm) )</td>
<td>( 25.415 )</td>
</tr>
<tr>
<td>( h ) ( (cm) )</td>
<td>( 0.07112 )</td>
</tr>
<tr>
<td>( T_{\text{init}} ) ( (K) )</td>
<td>( 220 )</td>
</tr>
</tbody>
</table>

Note, \( \alpha \) and \( E \) are functions of temperature [125].
Table 7.8: Numerical properties of thermal and structural models of the 2-D shock impingement panel.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal time step (μs)</td>
<td>1000</td>
</tr>
<tr>
<td>Structural time step (μs)</td>
<td>100</td>
</tr>
<tr>
<td>Structural modes</td>
<td>15</td>
</tr>
<tr>
<td>Structural integration points</td>
<td>1001</td>
</tr>
<tr>
<td>Thermal elements through length</td>
<td>1000</td>
</tr>
<tr>
<td>Thermal elements through thickness</td>
<td>4</td>
</tr>
<tr>
<td>Thermal DOFs</td>
<td>5005</td>
</tr>
</tbody>
</table>

discretized in time using a second-order accurate Crank-Nicolson scheme.

The structure is modeled as a panel in cylindrical bending with von Kármán strains [129], shown in Eq. (6.1). The model includes thermal strains due to non-uniform temperature through the length and thickness of the panel, as well as chordwise varying, temperature-dependent elastic modulus and coefficient of thermal expansion. The equation of motion is discretized using Galerkin’s method to replace the spatial dependence with a summation of fifteen free vibration mode-shapes. The equations of motion are discretized in time using the Newmark-β time integration scheme [129, 130]. A loosely coupled partitioned scheme is used to link the models which is second-order accurate in time [128].
7.4 Fluid-Thermal Analysis

A central aspect of this dissertation is the assumption of quasi-static flow conditions for the aerodynamic heating. In order to assess the validity of a quasi-static flow assumption, fully unsteady, quasi-static, and steady flow are considered for a fluid-thermal analysis of the 2-D panel undergoing forced vibration and subject to shock-turbulent boundary layer interactions. The fluid time step is set to 0.1ms and the thermal time step is set to 1.0ms. Also included in this analysis are two surrogate models, the corrected pointwise isothermal model (“CIM”) and the polynomial temperature basis model (“Poly”). The aerodynamic heating portion of these models is quasi-static. The aerodynamic pressure portion of the models uses the local piston theory correction where $T_{loc}$ is approximated at the boundary layer edge (“LPT$_{BL}$”). Note, the “CIM” uses the “Iso” model for the steady-state surrogate portion of the pressure. All of the cases with their respective aerodynamic heating and pressure sub-models are listed in Table 7.9.

In order to quantify the accuracy of the different approaches, comparisons are made in terms of the spatially averaged through-thickness temperature rise in the panel and the generalized aerodynamic forces (GAFs). The GAFs are computed using Eq. (5.18). For this analysis, the flow properties listed in Table 2.5, and the thermal and structural properties listed in Table 7.7 are used. Additionally, Eq. (2.5) defines the forced motion of the shock generator at 10 Hz, and Eq. (7.2) defines the
Table 7.9: Aerodynamic heating and pressure sub-models for the fluid-thermal-structural response of the HSV panel.

<table>
<thead>
<tr>
<th>Case</th>
<th>Aerodynamic Heating</th>
<th>Aerodynamic Pressure</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unsteady</td>
<td>Unsteady CFD</td>
<td>Unsteady CFD</td>
</tr>
<tr>
<td>Quasi–Static</td>
<td>Quasi–Static CFD</td>
<td>Quasi–Static CFD</td>
</tr>
<tr>
<td>Steady</td>
<td>Steady–State CFD</td>
<td>Steady–State CFD</td>
</tr>
<tr>
<td>CIM</td>
<td>CIM</td>
<td>Iso–LPT&lt;sub&gt;BL&lt;/sub&gt;</td>
</tr>
<tr>
<td>Poly</td>
<td>Poly</td>
<td>Poly–LPT&lt;sub&gt;BL&lt;/sub&gt;</td>
</tr>
</tbody>
</table>

forced motion of the panel in its first mode shape at 50 Hz.

\[
w(x, y, t) = 3 h \sin(2 \pi 50 t) \phi_1(x, y) \tag{7.2}
\]

Note, the shock generator angle varies ±3° about a nominal 10° wedge, and the panel varies ± 3 panel thicknesses (\(h\)) about its undeformed (i.e., flat) position. Comparisons of the spatially averaged through-thickness temperature rise in the panel at 2 seconds, and from 0 to 2 seconds at the 45% chord are shown in Fig. 7.11(a,b). The 45% chord location is chosen because this location yielded the greatest difference between the unsteady and steady results. Comparisons of the GAFs from 0 to 2 seconds are shown in Fig. 7.12(a), and from 1.9 to 2.0 seconds, in Fig. 7.12(b).

The mean and maximum percent error, as well as the overall maximum dimensional error of the quasi-static, steady, and surrogate cases are computed relative to the unsteady case in Table 7.10 using Eqs. (3.22) to (3.25). The results in Fig. 7.11 and Table 7.10 demonstrate that quasi-static aeroheating loads are sufficient for
Figure 7.11: Spatially averaged through-thickness temperature rise for the unsteady, quasi-static, steady, and surrogate models.
Figure 7.12: Generalized aerodynamic forces for the unsteady, quasi-static, steady, and surrogate models.
fluid-thermal-structural analysis of this 2-D panel configuration. The mean and max errors of the temperature loading after two seconds for the quasi-static and “CIM” cases are both under 1.5% and 0.8 K. In terms of the pressure loading, from Fig. 7.12(a,b) and Table 7.10 the quasi-static and “CIM” cases are within mean and max errors of 2.8% and 5.5%, respectively. Note these errors are primarily due to time lag effects, and the error in maximum and minimum loading for both the quasi-static and “CIM” cases compared to the unsteady model is under 1.5%. The quasi-static model assumption, which neglects unsteady effects on the pressure, did not result in significant errors for this case. This is presumably due to the shock impingement on the panel, which dominates the pressure load and is apparently readily captured using a quasi-static analysis due to the slow motion of the shock (10 Hz). Note that as expected, both the steady model and the “Poly” model yield poor results. The steady model predicts errors in the temperature rise and GAFs as high as 34% and 42%, respectively. The “Poly” model which was shown to have large errors in both aerodynamic heating and pressure in previous sections, predicts errors in the temperature rise and GAFs as high as 53% and 29%, respectively.
Table 7.10: Percent differences in the spatially averaged through-thickness temperature rise and generalized aerodynamic forces for several cases compared to the unsteady case for 2.0 seconds of response.

<table>
<thead>
<tr>
<th>Method</th>
<th>Temperature Rise</th>
<th>Generalized Aerodynamic Forces</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Avg. (%)</td>
<td>Max (%)</td>
</tr>
<tr>
<td>Quasi – Static</td>
<td>1.34</td>
<td>3.57</td>
</tr>
<tr>
<td>Steady</td>
<td>4.48</td>
<td>33.9</td>
</tr>
<tr>
<td>Poly</td>
<td>34.2</td>
<td>53.1</td>
</tr>
<tr>
<td>CIM</td>
<td><strong>0.971</strong></td>
<td>4.10</td>
</tr>
</tbody>
</table>
7.5 Fluid-Thermal-Structural Analysis

The 2-D shock impingement configuration is used to investigate F-T-S interactions of a compliant panel undergoing forced sinusoidally oscillating shock impingement. Several cases are considered in order to assess the impact of including and neglecting specific effects in a coupled F-T-S analysis. The cases and descriptions are listed in Table 7.11. First, several cases based on the “CIM” are constructed: 1) the base “Coupled” case, which is a coupled F-T-S simulation using the most accurate surrogate developed to compute the pressure and heat loads, 2) shock movement is neglected for both the aerodynamic heating and pressure, but panel deformation is included (“Def. Coupled”), and 3) panel deformation is neglected but shock movement is included (“Shock Coupled”). Next, two cases using the “Linear” aerodynamic heating model are considered: 1) the base “Linear” model, with the same pressure model as the “Coupled” case (“Linear”), and 2) an uncoupled model which uses the “Linear” model for heating and a constant pressure based on the initial conditions of a 10° shock generator angle and a flat panel (“Uncoupled”). Finally, polynomial surface temperature surrogate pressure and heating models are considered (“Poly”).

An important consideration for these cases is the pressure acting on the back of the panel (i.e., the side of the panel not subject to supersonic flow). A low backpressure will bias the panel to always buckle out of the supersonic flow due to shock impingement, and a high backpressure will buckle the panel into the flow. Thus, as
Table 7.11: Descriptions of the fluid-thermal-structural cases for the 2-D shock impingement panel.

<table>
<thead>
<tr>
<th>Case</th>
<th>Aerodynamic Heating</th>
<th>Aerodynamic Pressure</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coupled</td>
<td>CIM</td>
<td>ISO–LPT&lt;sub&gt;BL&lt;/sub&gt;</td>
</tr>
<tr>
<td>Def. Coupled</td>
<td>CIM</td>
<td>ISO–LPT&lt;sub&gt;BL&lt;/sub&gt;</td>
</tr>
<tr>
<td></td>
<td>Neglects shock movement</td>
<td>Neglects shock movement</td>
</tr>
<tr>
<td>Shock Coupled</td>
<td>CIM</td>
<td>ISO–LPT&lt;sub&gt;BL&lt;/sub&gt;</td>
</tr>
<tr>
<td></td>
<td>Neglects panel deformation</td>
<td>Neglects panel deformation</td>
</tr>
<tr>
<td>Linear</td>
<td>Linear</td>
<td>ISO–LPT&lt;sub&gt;BL&lt;/sub&gt;</td>
</tr>
<tr>
<td></td>
<td>Neglects all deformation</td>
<td></td>
</tr>
<tr>
<td>Uncoupled</td>
<td>Linear</td>
<td>Constant using initial conditions</td>
</tr>
<tr>
<td></td>
<td>Neglects all deformation</td>
<td></td>
</tr>
<tr>
<td>Poly</td>
<td>Poly</td>
<td>Poly–LPT&lt;sub&gt;BL&lt;/sub&gt;</td>
</tr>
</tbody>
</table>

a first assessment, a range of backpressures are considered in order to identify the critical backpressure that causes the panel to switch from buckling out of the flow to buckling into the flow. Then, the performance of all of the models relative to the same backpressure is considered, and 120 seconds of fluid-thermal-structural response is computed.

7.5.1 Effect of Backpressure on Buckling Direction

Panel buckling direction can have significant effects on the transient behavior of the panel and the aerodynamics, including changes in aerodynamic loading, panel temperature, and the stability of the panel against aeroelastic flutter phenomenon.
Thus, the critical backpressure that determines the buckling direction for all of the models is investigated. The values relative to the free stream pressure are listed in Table 7.12 ($P_\infty = 2970\text{Pa}$). The only case which has a lower critical backpressure than the “Coupled” case is the “Linear” case, where the only variation between these cases is the aerodynamic heating model. By neglecting shock movement (“Def. Coupled”) the critical backpressure is increased 4.2%, and by neglecting feedback of deformation but including shock movement (“Shock Coupled”) the critical backpressure is increased 6.3%. The “Uncoupled” model which neglects all movement and uses the linear aerodynamic heating model predicts a 5.2% rise in the critical backpressure relative to the “Coupled” case. The “Poly” case predicts the highest rise in critical backpressure at 8.4%.

Table 7.12: Comparison of the critical backpressure which causes the panel to switch from buckling out of the flow to buckling into the flow ($P_\infty = 2970\text{Pa}$).

<table>
<thead>
<tr>
<th>Case</th>
<th>Critical Backpressure</th>
<th>Difference from Coupled</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coupled</td>
<td>$1.61\times P_\infty$</td>
<td>–</td>
</tr>
<tr>
<td>Def. Coupled</td>
<td>$1.67\times P_\infty$</td>
<td>4.17%</td>
</tr>
<tr>
<td>Shock Coupled</td>
<td>$1.71\times P_\infty$</td>
<td>6.29%</td>
</tr>
<tr>
<td>Linear</td>
<td>$1.58\times P_\infty$</td>
<td>-1.56%</td>
</tr>
<tr>
<td>Uncoupled</td>
<td>$1.69\times P_\infty$</td>
<td>5.23%</td>
</tr>
<tr>
<td>Poly</td>
<td>$1.74\times P_\infty$</td>
<td>8.41%</td>
</tr>
</tbody>
</table>
7.5.2 Fluid-Thermal-Structural Response

For comparison, 120 seconds of fluid-thermal-structural response with a backpressure 2000 Pa above free stream \((1.67 \times P_{\infty})\) is considered for all of the cases listed in Table 7.12. This backpressure will result in the “Shock Coupled” and “Uncoupled” models buckling out of the flow, while the other models will buckle into the flow. The structural response is shown in Fig. 7.13 and the thermal response is shown in Fig. 7.14, for all of the models except for the “Poly” case. The results for the “Poly” model are discussed at the end of this chapter, due to unique results for this model. Note, the structural response is represented by panel deformation. The entire 120 second response at the mid-chord is shown in Fig. 7.13(a); two snapshots of the deformation at 5 and 120 seconds are shown in (b); the mid-chord displacement for the last second of response is shown in (c) and (d); and the first 4 seconds of response in (e). Note, positive displacement represents displacement into the flow-field, while negative displacement is out of the flow-field.

For the cases which included the shock movement in the pressure model, “Coupled”, “Shock Coupled”, and “Linear”, both the last second of response (c,d) and the first 4 seconds of response (e) show oscillations of the mid-chord displacement. In the last second of response, the two models which buckled into the flow, “Linear” and “Coupled”, show oscillations approximately five times larger than for the case which buckled out of the flow, “Shock Coupled”. For the first three seconds of response, snap through oscillations up to ±4 thicknesses are observed, Fig. 7.13(e).
The frequency of the snap through occurs at 10 Hz, and is driven by the 10 Hz forced motion of the shock generator, which is not included in either the “Uncoupled” or “Def. Coupled” analyses. The ceasing of the snap through event after a few seconds is due to panel stiffening during post-buckling as a result of the increasing in-plane thermal force. A similar panel stiffening during post-buckling was observed in the Mach 7.5 results for the F-T-S analysis of the HSV panel.

For the thermal response, Fig. 7.14, the average through thickness temperature rise from 0 to 120 seconds is shown in (a), and the temperature rise across the plate at 5 and 120 seconds is shown in (b). The average temperature is roughly equivalent for all of the cases, however the “Coupled” case begins to diverge from the other methods between 60 and 80 seconds, and by 120 seconds predicts approximately a 20 K or 6% lower average temperature. The average temperature in this configuration drives the displacement, thus for the other four cases the displacements are roughly the same magnitude, while for the “Coupled” case the displacements are 4% lower on average from 119 to 120 seconds, shown in Fig. 7.13(d). The temperature profiles also vary between these cases, Fig. 7.14(b), where the “Coupled” and “Def. Coupled” show similar profile shapes at 120 seconds, while all of the other models which neglect deformation have similar temperature profiles to each other, but different from the previous two cases. These results illustrate that neglecting the movement of the shock impacts possible snap through and subsequent oscillations in the structural response, while the cases which neglected the effect of deformation on the pressure predicted the wrong buckling direction, and
(a) Displacement at the 50% chord.

(b) Displacement profiles at 5 and 120 seconds.

(c) Displacement at the 50% chord 119 to 120 seconds.

(d) Displacement at the 50% chord 119 to 120 seconds.

(e) Displacement at the 50% chord over the first 4 seconds.

Figure 7.13: Structural response for a backpressure 2000 Pa above free stream \((1.673 \times P_\infty)\) for the various models. Free stream pressure = 2970 Pa.
all of the cases predicted average thermal responses at least 6% different from the “Coupled” case at 120 seconds.

![Graph showing average temperature rise through the plate.](image1)

(a) Average temperature rise through the plate.

![Graph showing temperature rise across the plate at 5 and 120 seconds.](image2)

(b) Temperature rise across the plate at 5 and 120 seconds.

Figure 7.14: Thermal response for a backpressure 2000 Pa above free stream \((1.673 \times P_{\infty})\) for the various models. Free stream pressure = 2970 Pa.

Finally, the “Poly” case responses are shown in Fig. 7.15 for two different back-
pressures; one which buckles the panel into the flow (1.774×P∞) and one which
does not (1.766×P∞). The thermal response is almost identical for both of these
backpressures (e,f), and the displacements are also similar in magnitude, though
opposite in direction. Note, the temperature rise for this panel exceeded the ma-
terial limits listed in Ref. [125] at approximately 90 seconds. Therefore at 90 sec-
sons the panel response becomes chaotic and the simulation results are no longer
meaningful. Note, as in the previous aerodynamic heating and pressure analysis
with the “Poly” model, this simulation exceeded the bounds of this model both
in deformation and temperature profile limits, resulting in poor predictions of the
aerothermodynamic loads.
(a) Displacement at the 50% chord.

(b) Displacement at the 50% chord 89 to 90 seconds.

(c) Displacement at the 50% chord over the first 4 seconds.

(d) Displacement profiles at 5 and 90 seconds.

(e) Average temperature rise through the plate.

(f) Temperature rise across the plate at 5 and 90 seconds.

Figure 7.15: Structural and thermal response for several backpressures for the “Poly” model. Free stream pressure = 2970 Pa.

218
7.6 Three-Dimensional Aerodynamic Heat Flux Modeling

A three dimensional representation of the previous configuration is also considered, based on the 3-D shock impingement panel discussed in Section 2.4. Three different configurations are considered for the three-dimensional analysis: (1) a flat surface without shock impingement, (2) variable 3-D deformation without shock impingement, and (3) variable 3-D deformation with variable shock impingement location. Aerodynamic heating models for these configurations are discussed next.

7.6.1 Neglecting Deformation and Shock Impingement

The first three dimensional analysis is performed on a flat surface with variable surface temperatures in both $x$ and $y$. The flow conditions considered are the Mach 3.0 flow conditions listed in Table 2.7. Similar to the previous analyses, several surrogate models are constructed and verified for this configuration, followed by comparison of the models to 50 CFD test cases with variable surface temperatures.

Construction and Verification of CFD Surrogates

Only three aerodynamic heating models are considered: the heat transfer coefficient model which produces a linear heat flux with respect to surface temperature (“Linear”), the pointwise surface temperature model (“Iso”), and the corrected
pointwise model ("CIM").

**Linearized Model**

Similar to the 2-D deformable panel subject to shock impingement, the 3-D "Linear" model is computed directly from 2 CFD training solutions: (1) free stream surface temperature and (2) adiabatic surface temperature. The heat transfer coefficient is computed using Eq. (4.4), and the aerodynamic heating is computed using Eq. (4.5).

**Pointwise Model**

The pointwise surrogate ("Iso") is constructed from 10 CFD snap shots, where the only parameter varied is the isothermal surface temperature between the bounds listed in Table 2.7. Verification of the surrogate for prediction of aerodynamic heat flux is conducted relative to 10 full-order CFD solutions with variable isothermal surface temperatures. Note, similar to the flat 2-D analysis, due to nearly constant, small magnitude values of the heat flux from several of the full-order isothermal CFD solutions, the normalized errors are unrepresentative of the quality of the model. A comparison of the maximum dimensional error to the peak heat flux results in a maximum difference of 1.3% for turbulent flow.
Corrected Pointwise Isothermal Model

The three-dimensional correction for the “CIM”, $\Delta q$, is applied in 2-D flow-wise strips using Eq. (4.32), where $x$ should now be in the direction of the free stream flow. The “CIM” is constructed in the same fashion as before, using the 3-D “Iso” model. As before, no additional surrogate verification is necessary for this model.

Comparison of Aerodynamic Heating Models

For comparison of the aerodynamic heating predictions, 50 full-order CFD turbulent test cases are computed, which vary only the surface temperature. In order to create arbitrary surface temperature profiles for the 3-D analysis, the surface temperature is represented by:

$$T_w(x, y) = T_0 + \sum_{i=1}^{6} T_i \Phi_i(x, y)$$

(7.3)

where $T_0$ represents a uniform temperature over the entire surface, and $\Phi_i(x, y)$ are the first 6 structural free vibration modeshepes, normalized between 0 and 1. The temperature modal amplitudes ($T_i$) are varied independently relative to the structural modal amplitudes ($\alpha_i^*$) in the generation of test cases. The bounds of $T_0$ are the same as the bounds on $T_w$ from Table 2.7, and the bounds of $T_1-T_6$ are $\pm 100$. The 50 test cases for this analysis are chosen through LHS sampling of the 7 surface temperature profile amplitudes, $T_i$. Results are listed in Table 7.13, which follow the trends of the 2-D flat panel analysis and again highlight the excellent performance of the “CIM” model.
Table 7.13: Error in the aerodynamic heat flux for several models for the 3-D panel without deformation or shock impingement compared to CFD.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (W/cm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>12.5</td>
<td>71.5</td>
<td>0.54</td>
</tr>
<tr>
<td>Iso</td>
<td>6.79</td>
<td>62.9</td>
<td>0.42</td>
</tr>
<tr>
<td>CIM</td>
<td>1.71</td>
<td>19.1</td>
<td>0.13</td>
</tr>
</tbody>
</table>

In order to gain further insight into the sources of error in the present problem, consider the mean spatial error shown in Fig. 7.16 for the three models. This is computed using:

$$E^* (%) = \frac{1}{n} \sum_{j=1}^{n} \frac{|\text{MODEL}_j - \text{CFD}_j|}{\text{Norm}_j}$$  

(7.4)

where $n$ corresponds to the total number test cases and $j$ corresponds to the $j^{th}$ test case. Both the mean percent and mean dimensional errors are presented as a function of $x$ and $y$ for all three models. All three models have the highest error along the mid-span of the panel. The “Linear” model has mean errors exceeding 15% over the surface and an overall mean of 12.5%. The “Iso” model preforms somewhat better, mean errors still exceed 15%, but the overall mean error is reduced to 6.8%. However, the “CIM” outperforms both other models with a mean errors consistently under 5% over the surface and an overall mean error of 1.7%.

The specific test case with the maximum dimensional error for the “CIM” is shown in Fig. 7.17 for visual comparison of the models. The temperature profile for this case is shown in Fig. 7.17(a) along with the mid-span heat flux in Fig. 7.17(b). The actual CFD heat flux on the surface is shown in Fig. 7.17(c), and
Figure 7.16: Mean error as a function of space ($E^*$) for several aerodynamic heating models compared to CFD for the 3-D flat panel analysis: $M_\infty = 3.0$, $Alt. = 24km$, $L = .254m$, $S = .127m$. 

(a) Mean Percent Error (“Linear”).  
(b) Mean Percent Error (“Iso”).  
(c) Mean Percent Error (“CIM”).
the heat flux for the "CIM" is shown in Fig. 7.17(d). For this particular test case, both the "Linear" and "Iso" models have similar levels of error, and even though this is the worst case for the "CIM", it still outperforms the other two models. Note, as shown in Fig. 7.17(d) the "CIM" is clearly capable of predicting the heat flux for a three-dimensional configuration.

Figure 7.17: Comparison of several turbulent aerodynamic heating predictions to the Benchmark (CFD) for the 3-D flat panel without shock impingement: \( M_\infty = 3.0 \), Alt. = 24km, \( L = .254m \), \( S = .127m \).
7.6.2 Including Deformation

The next analysis is performed with variable surface temperatures and deformations in both $x$ and $y$. The flow conditions considered are the Mach 3.0 flow conditions listed in Table 2.7. Similar to the previous analyses, several surrogate models are constructed and verified for this configuration, followed by comparison of the models to 50 CFD test cases with variable surface temperatures and surface deformations.

Construction and Verification of CFD Surrogates

Again, only three aerodynamic heating models are considered: the heat transfer coefficient model which produces a linear heat flux with respect to surface temperature (“Linear”), the pointwise surface temperature model (“Iso”), and the corrected pointwise model (“CIM”).

Linearized Model

The “Linear” model neglects deformation, thus it is identical to the previous 3-D “Linear” model, and is constructed from two sample points: (1) surface temperature set to free stream and (2) surface temperature set to the adiabatic temperature. The heat transfer coefficient is computed using Eq. (4.4), and the aerodynamic heating is computed using Eq. (4.5).
**Pointwise Model**

The pointwise surrogate ("Iso") is constructed using kriging from the output of 200 CFD snap shots, where the parameters varied are: (1) isothermal surface temperature and (2–7) first six modal amplitudes \(a_1^*–a_6^*\) between the bounds listed in Table 2.7. Verification of the surrogate for prediction of aerodynamic heat flux is conducted relative to 10 full-order CFD solutions with variable isothermal surface temperatures and deformations. The resulting mean error is 4.71%.

**Corrected Pointwise Isothermal Model**

Similar to the previous three-dimensional analysis using the “CIM”, \(\Delta q\) is applied in 2-D flow-wise strips using Eq. (4.32). The “CIM” is constructed in the same fashion as before, using this “Iso” model. Note, the effects of deformation are assumed to be small in the correction, thus free stream properties are used in place of the boundary layer edge properties in Eq. (4.33). As before, no additional surrogate verification is necessary for this model.

**Comparison of Aerodynamic Heating Models**

These models are compared to 50 turbulent test cases which vary both the surface temperature and surface deformation. The variable surface temperature is determined in the same manner as the 3-D analysis which neglected deformation. The comparisons between the 50 test cases and the models are shown in Table 7.14 and...
Fig. 7.18. Similar to the 3-D flat analysis, all three models show the highest error along the mid-span of the panel, Fig. 7.18. The “Linear” model performs the worst with mean errors exceeding 25%. The “Iso” model provides an improvement since it is capable of capturing the effect of deformation, with overall errors of 7.1% for the mean. However, the “CIM” again clearly yields the best approximation for the heat flux, shown in Fig. 7.18 (e) and (f), with the lowest mean and maximum errors.

Table 7.14: Error in the aerodynamic heat flux for several models for the 3-D panel without shock impingement compared to CFD.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (W/cm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>21.3</td>
<td>146</td>
<td>1.75</td>
</tr>
<tr>
<td>Iso</td>
<td>7.12</td>
<td>80.5</td>
<td>0.54</td>
</tr>
<tr>
<td>CIM</td>
<td>2.76</td>
<td>42.7</td>
<td>0.26</td>
</tr>
</tbody>
</table>

The test case with the maximum dimensional error for the “CIM” is shown in Fig. 7.19. The deformation profile is shown in Fig. 7.19(a), with the temperature contour overlayed. The mid-span heat flux results of the models and CFD (Benchmark) are shown in Fig. 7.19(b). The three-dimensional heat flux from CFD is shown in Fig. 7.19(c), and the heat flux for the “CIM” is shown in Fig. 7.19(d). The large error for the “Linear” model, Fig. 7.19(b), is due to neglecting the effect of deformation. For the “Iso” model the effect of neglecting surface temperature gradients is also visible in Fig. 7.19(b). The “CIM” provides a good approximation, even though this is the worst case out of 50, and is clearly capable of handling complex three-dimensional flows and the resulting heat flux distributions Fig. 7.19(d).
Figure 7.18: Mean error as a function of space ($E^*$) for several aerodynamic heating models compared to CFD for the 3-D deformable panel analysis: $M_\infty = 3.0$, $Alt. = 24km$, $L = .254m$, $S = .127m$. 

(a) Mean Percent Error (“Linear”). 

(b) Mean Percent Error (“Iso”). 

(c) Mean Percent Error (“CIM”).
(a) Temperature and Deformation Profiles: $a_1^*-a_6^* = -8.01, -2.51, 4.14, -1.86, 2.25, -1.62$.

(b) Mid-Span Heat Flux.

(c) Benchmark Heat Flux (CFD).

(d) “CIM” Heat Flux.

Figure 7.19: Comparison of several turbulent aerodynamic heating predictions to the Benchmark (CFD) for the 3-D deformable panel without shock impingement: $M_\infty = 3.0$, $Alt. = 24km$, $L = .254m$, $S = .127m$, $h = 0.635mm$. 

229
7.6.3 Including Deformation and Shock Impingement

The last three-dimensional analysis is performed with variable surface temperatures, deformations, and shock impingement location. The flow conditions considered are the Mach 2.0 flow conditions listed in Table 2.7. The shock generator, shown in Figs. 2.7 through 2.9, does not extend across the entire span of the wind-tunnel. Thus, complex shock-shock and shock-boundary layer interactions are expected due to the initial oblique shock on the generator and reflected shocks off of the panel and side-walls. Similar to the previous analyses, several surrogate models are constructed and verified for this configuration, followed by comparison of the models to 50 CFD test cases with variable surface temperatures, surface deformations, and shock impingement locations.

Construction and Verification of CFD Surrogates

The aerodynamic heating models considered for the 3-D analysis are: the “Linear” model, “Iso” model, and the “CIM”.

Linearized Model

Similar to the 2-D deformable panel subject to shock impingement, the 3-D “Linear” model is computed directly from 2 CFD training solutions: (1) free stream surface temperature and (2) adiabatic surface temperature. As before, the shock generator angle is held constant at 10° in both of the training solutions. The heat transfer
coefficient is computed using Eq. (4.4), and the aerodynamic heating is computed using Eq. (4.5).

**Pointwise Model**

The pointwise surrogate (“Iso”) is constructed from 1500 CFD training solutions in the same manner as for the 2-D shock impingement analysis. Verification of the surrogate for prediction of aerodynamic heat flux is conducted relative to 15 full-order CFD solutions with variable isothermal surface temperatures, deformations, and shock impingement locations. The resulting mean error is 4.13%. Similar to the 2-D results, the maximum errors are large and unrepresentative of the quality of the model due to small differences in the predicted shock impingement location.

**Corrected Pointwise Isothermal Model**

The corrected pointwise isothermal model (“CIM”) is constructed in the same manner as for the 2-D shock impingement analysis. Verification of the pointwise surrogates for predicting density, temperature, and velocity at the edge of the boundary layer is carried out through comparison with 15 full-order CFD test cases with variable isothermal surface temperatures, deformations, and shock impingement locations. The surrogate models yield mean errors of 3.01% for density, 1.26% for temperature, and 0.98% for velocity.
Comparison of Aerodynamic Heating Models

For comparison of the aerodynamic heating predictions, 50 full-order CFD turbulent test cases are computed. The 50 test cases are chosen through LHS sampling of the parameters: surface temperature profile amplitudes, $T_i$; surface deformation modal amplitudes, $a_i^*$; and shock generator angle. Results are listed in Table 7.15, which follow the trends of the previous analyses and again highlight the excellent performance of the “CIM” model. The mean spatial error is shown in Fig. 7.20 for the three models. The highest error for the “Linear” model occurs near the 10 to 20% chord and near the trailing edge of the panel, which corresponds with shock impingement locations. The pointwise “Iso” model, which accounts for both deformation and variable shock impingement location, provides a significant improvement, with mean errors generally under 20% over the entire surface. However, the “CIM” represents a clear improvement by correcting for gradients in the spatial surface temperature, yielding mean errors generally less than 5% over the entire surface.

Table 7.15: Error in the aerodynamic heat flux for several models for the 3-D deformable panel subject to shock impingement compared to CFD.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error ($W/cm^2$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>29.0</td>
<td>227</td>
<td>7.41</td>
</tr>
<tr>
<td>Iso</td>
<td>13.2</td>
<td>108</td>
<td>2.60</td>
</tr>
<tr>
<td>CIM</td>
<td><strong>5.05</strong></td>
<td><strong>74.0</strong></td>
<td><strong>2.29</strong></td>
</tr>
</tbody>
</table>
Figure 7.20: Mean error as a function of space ($E^*$) for several aerodynamic heating models compared to CFD for the 3-D deformable panel subject to shock impingement: $M_\infty = 2.0$, $\rho_\infty = 0.713 \text{ kg/m}^3$, $T_\infty = 215.5 \text{ K}$, $L = .254m$, $S = .127m$. 
For further analysis, consider Fig. 7.21, which shows a comparison of the mid-span heat flux for the test case corresponding with the highest dimensional error of the “CIM”. The deformation profile, with the temperature contour overlaid, is shown in Fig. 7.21(a). The impact of a spatially variable surface temperature on the heat flux is evident by the difference between the “Iso” and “CIM” results for this case. In terms of the peak heat flux, the “CIM” is in excellent agreement with the CFD prediction, within 0.3 \( W/cm^2 \) or 2.1%; while the “Iso” model under predicts the peak heat flux by 1.7 \( W/cm^2 \) or 13.8%.
(a) Temperature and Deformation Profiles:
\[a_1 - a_6 = -8.85, 1.67, 1.21, -0.67, 0.08, 1.47.\]

(b) Mid-Span Heat Flux.

(c) Benchmark Heat Flux (CFD).

(d) “CIM” Heat Flux.

(e) “Linear” Heat Flux.

(f) “Iso” Heat Flux.

Figure 7.21: Comparison of several turbulent aerodynamic heating predictions to the Benchmark (CFD) for the 3-D deformable panel subject to shock impingement: Shock Generator=7.67°, \(M_\infty = 2.0, \rho_\infty = 0.713 \text{ kg/m}^3, T_\infty = 215.5 \text{ K}, L = .254m, S = .127m, h = 0.635mm.\)
7.7 Three-Dimensional Unsteady Pressure Modeling with Shock Impingement

Unsteady aerodynamic pressure for three-dimensional surface motion with oscillating shock impingement is considered next. Similar to the 2-D oscillating shock impingement analysis, only one case is considered; oscillations of the first clamped-clamped panel mode at 50 Hz at a non-dimensional amplitude of 5. The shock-generator oscillates $\pm 2^\circ$ about a $10^\circ$ nominal angle at 10 Hz. The free stream properties are held constant using the Mach 2.0 flow conditions listed in Table 2.7. GAFs are computed for the unsteady Navier-Stokes (“NS”), Euler (“EU”), second- and third-order piston theory (“PT2” and “PT3”), steady-state Navier-Stokes (“NSss”), and several corrections for the steady-state Navier-Stokes using piston theory and local piston theory. The errors relative to the unsteady Navier-Stokes solution are listed in Table 7.16.

The GAFs for several of the models are presented in Fig. 7.22, where (a) and (c) are the full GAFs, and (b) and (d) are the unsteady components. For all of these cases, the unsteady components of the GAFs are approximately two-orders of magnitude smaller than the overall GAFs. Thus, the steady-state models have small error relative to their unsteady counterparts, less than 2% maximum error. However, there is a clear difference in the GAFs between analytical, Euler, and Navier-Stokes modeling of this problem, shown in Figs. 7.22 (a) and (b), where
Table 7.16: Error in GAFs for an oscillating shock impinging on a vibrating 3-D panel, comparing Navier-Stokes, analytical, and Euler based methods for oscillations of the first panel mode.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EU</td>
<td>13.4</td>
<td>22.6</td>
<td>25.8</td>
</tr>
<tr>
<td>PT^2</td>
<td>99.2</td>
<td>116</td>
<td>133</td>
</tr>
<tr>
<td>PT^3</td>
<td>99.2</td>
<td>116</td>
<td>133</td>
</tr>
<tr>
<td>NS_{SS}</td>
<td>1.60</td>
<td>4.04</td>
<td>4.62</td>
</tr>
<tr>
<td>NS_{SS}-PT^2</td>
<td>1.27</td>
<td><strong>3.16</strong></td>
<td><strong>3.62</strong></td>
</tr>
<tr>
<td>NS_{SS}-PT^3</td>
<td>1.27</td>
<td><strong>3.16</strong></td>
<td><strong>3.62</strong></td>
</tr>
<tr>
<td>NS_{SS}-LPT_{BL}</td>
<td>1.23</td>
<td><strong>3.16</strong></td>
<td><strong>3.62</strong></td>
</tr>
<tr>
<td>NS_{SS}-LPT_{EU}</td>
<td>1.23</td>
<td><strong>3.16</strong></td>
<td><strong>3.62</strong></td>
</tr>
<tr>
<td>NS_{SS}-LPT_{T\infty}</td>
<td><strong>1.22</strong></td>
<td>3.17</td>
<td>3.63</td>
</tr>
</tbody>
</table>

Mean errors are approximately 13% for Euler and almost 100% for classical piston theory. Note that the full GAFs for the classical piston theory methods do not appear in (a) because of the large error. Clearly as with the 2-D shock-impingement analysis, the dominant feature of this problem is not the unsteady motion of the panel surface, but the inclusion of shock impingement. The analytical methods do not capture the shock impingement and the Euler based models do not capture it accurately relative to Navier-Stokes.

The “EU” and “EU_{SS}” methods compare favorably to each other, indicating the unsteady component is small relative to the overall GAFs. The unsteady components from the classical piston theory models also match up well with the unsteady component of Euler. Figures 7.22 (c) and (d) are a comparison of two of
the correction methods for the steady-state Navier-Stokes model. Note, based on
the errors in Table 7.16 all of the corrected models predict roughly the same values
for the GAFs, thus one piston theory correction and one local piston theory cor-
rection are shown. Though, all of the corrected models have low error relative to
Navier-Stokes, the unsteady components of the Navier-Stokes solution are clearly
different.

Figure 7.22: Generalized Aerodynamic Forces (GAFs) for an oscillating shock im-
pinging on a vibrating 3-D panel over 5 panel cycles. Shock generator: \( \pm 2^\circ \) about
a nominal 10\(^{\circ}\) angle at 10 Hz. Panel: \( a_1 = 5, f = 50 \) Hz. \( M_\infty = 2.0, T_\infty = 215.5 \) K,
\( \rho_\infty = 0.7127 \text{kg/m}^3, T_w = 215.5 \) K.
Next, the steady-state pointwise “Iso” surrogate model is introduced for this problem. Note, three-dimensional unsteady pressure using the “Poly” model is considered for the 3-D control surface in Section 5.3. Therefore, only the pointwise model is considered here. The parameter space, 1500 CFD training snap shots, and development of the surrogate model for aerodynamic heating is discussed in Section 7.6.3. The steady-state aerodynamic pressure surrogate is developed from the same snap shots and input parameters as for the aerodynamic heating surrogate. This model along with the unsteady corrections are compared with the unsteady Navier-Stokes solution in Table 7.17 and Fig. 7.23. The steady-state “Iso” model has similar errors as the steady-state Navier-Stokes model, approximately 4% maximum error. When including the unsteady corrections, the “Iso”-based models have consistently low errors, approximately 1.5% on average and 3% maximum for all cases. Note, for the local piston theory using local temperatures at the approximate boundary layer edge (“Iso–LPT_{BL}”), the local temperature is computed from a surrogate using the 1500 CFD training snap shots.
Figure 7.23: Surrogate Generalized Aerodynamic Forces (GAFs) for an oscillating shock impinging on a vibrating 3-D panel over 5 panel cycles. Shock generator: ± 2° about a nominal 10° angle at 10 Hz. Panel: $a_1^* = 5$, $f = 50$ Hz. $M_{\infty} = 2.0$, $T_{\infty} = 215.5$ K, $\rho_{\infty} = 0.7127$ kg/m$^3$, $T_w = 215.5$ K.
Table 7.17: Error in GAFs for an oscillating shock impinging on a vibrating 3-D panel, comparing Navier-Stokes and surrogate-based methods for oscillations of the first panel mode.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mean Error (%)</th>
<th>Max Error (%)</th>
<th>Max Error (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Iso</td>
<td>1.70</td>
<td>3.99</td>
<td>4.57</td>
</tr>
<tr>
<td>Iso–PT²</td>
<td>1.46</td>
<td>3.07</td>
<td>3.52</td>
</tr>
<tr>
<td>Iso–PT³</td>
<td>1.46</td>
<td>3.07</td>
<td>3.52</td>
</tr>
<tr>
<td>Iso–LPT_{BL}</td>
<td>1.45</td>
<td>3.12</td>
<td>3.57</td>
</tr>
<tr>
<td>Iso–LPT_{EU}</td>
<td>1.45</td>
<td>3.12</td>
<td>3.57</td>
</tr>
<tr>
<td>Iso–LPT_{T∞}</td>
<td><strong>1.44</strong></td>
<td>3.14</td>
<td>3.59</td>
</tr>
</tbody>
</table>
Chapter 8

Computational Requirements

The computational requirements associated with development and implementation of analytical, surrogate, and full-order CFD aero thermodynamic loads prediction are assessed in this chapter. The models are considered in the context of fluid-thermal-structural analysis. Results are presented for the 2-D cases using the fluid-thermal-structural simulations discussed in the previous two chapters. Then the cost of the 3-D models is examined.

8.1 Two-Dimensional Fluid-Thermal Structural Simulation

First, the computational cost of the HSV panel simulation conducted Chapter 6 is examined, then the cost of the 2-D shock impingement panel simulation from Chapter 7 is considered.
8.1.1 HSV Panel

In order to further compare the different modeling approaches considered in Chapter 6, this section discusses computational requirements for the analytical model ("C–M") [21], surrogate modeling, and full-order modeling in the context of fluid-thermal-structural simulation. The computational expense of these methods are compared in Table 8.1. The cases and their descriptions are listed in Table 6.6. Note that these computational times are based on a time-step size of 0.0005 sec for the fluid-structural (F-S) analysis, and 0.1 sec for the fluid-thermal (F-T) analysis [21]. The first column represents the computational time required to compute one iteration of the pressure and the heat flux. For the CFD analysis this number is approximated from the time required for one time step in an unsteady analysis, starting from a converged steady-state solution. One fluid-thermal-structural (F-T-S) cycle represents the time required to proceed through the entire model once (1/10th of a second of response). The final column represents the computational time required to produce 1500 seconds of fluid-thermal-structural response.

One F-T-S cycle for the surrogate methods takes approximately two and a half to four times as long as the theoretical model used in [21] due to differences in spatial integration of the surface pressure during the Runge-Kutta time marching of Eq. (6.1). Recall that surrogate based approaches require numerical integration of the pressure, while in [21] the GAFs were computed using an exact integration of the piston theory pressure prior to time-marching. Note, the thermal model, struc-
Table 8.1: Computational cost of different fluid-thermal-structural modeling approaches for the HSV panel.

<table>
<thead>
<tr>
<th>Model</th>
<th>1 Iteration of Aero Loads (seconds)</th>
<th>1 F-T-S Cycle (min)</th>
<th>1500 seconds of Response (hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>C–M&lt;sup&gt;a&lt;/sup&gt;</td>
<td>0.0845</td>
<td>0.081</td>
<td>20.30</td>
</tr>
<tr>
<td>CIM&lt;sup&gt;a&lt;/sup&gt;</td>
<td>0.0831</td>
<td>0.359</td>
<td>89.97</td>
</tr>
<tr>
<td>Linear&lt;sup&gt;a&lt;/sup&gt;</td>
<td>0.0596</td>
<td>0.307</td>
<td>76.89</td>
</tr>
<tr>
<td>Poly&lt;sup&gt;a&lt;/sup&gt;</td>
<td>0.0126</td>
<td>0.203</td>
<td>50.71</td>
</tr>
<tr>
<td>Full-Order CFD&lt;sup&gt;b&lt;/sup&gt;</td>
<td>35.94</td>
<td>120&lt;sup&gt;d&lt;/sup&gt;</td>
<td>30,000&lt;sup&gt;d&lt;/sup&gt;</td>
</tr>
</tbody>
</table>

<sup>a</sup> One 2.60 GHz Opteron processor core, 2.0 GB RAM.
<sup>b</sup> Thirty-two 2.60 GHz Opteron processor cores, 2.0 GB RAM each.
<sup>c</sup> 1/10th of a second of response: 1 F-T time step, 200 F-S time steps.
<sup>d</sup> Projected time.
tural model, interface, and integration accounts for 43.7 hrs of the response time for the surrogates. While the surrogate is more expensive relative to the theoretical aerothermodynamic approach in this context, it is still several orders of magnitude more efficient than a standard CFD solution. For the present analysis, the use of CFD within aerothermoelastic analysis is impractical for the response times shown in Fig. 6.12 (e.g., up to approximately 30 minutes of response). The “Poly” surrogate which produced poor results relative to the other methods, is almost twice as fast as the other surrogates. This is due to modeling the entire temperature using polynomials inside of the “Poly” surrogate, while for the other two surrogate methods, the pressure and heating is predicted in a point-wise method in order to account for surface temperature variations. Finally, note that the computational overhead required to generate the 1500 training snap shots for the “CIM” surrogate is around 30 minutes per snap shot on 32 processor cores. A benefit of this surrogate strategy is that all of the snap shots can all be generated in parallel, thus reducing the wall time drastically.

8.1.2 Two-Dimensional Shock Impingement Panel

The computational requirements for the different cases considered in Chapter 7 are compared in terms of the cost of 120 seconds fluid-thermal-structural response of the 2-D panel with shock-impingement. The cases included and descriptions of each case are listed in Table 7.11. Note, the “Def. Coupled” and “Shock Coupled” cases require use of the “CIM” surrogate, thus their computational times are identi-
The computational expense of the approaches are compared in Table 8.2. The computational times are based on a fluid-structural time-step of 0.0001 sec and a fluid-thermal time-step of 0.001 sec [128]. The first column represents the computational time required to generate one solution of the pressure and the heat flux. For the CFD analysis this number is approximated from the time required for one time step in an unsteady analysis, starting from a converged steady-state solution. The final column represents the computational time required to produce 120 seconds of fluid-thermal-structural response.

Table 8.2: Computational cost of different fluid-thermal-structural modeling approaches for the 2-D panel with shock impingement.

<table>
<thead>
<tr>
<th>Method</th>
<th>1 Iteration of Aero Loads (seconds)</th>
<th>120 seconds of Response (hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Uncoupled(^a)</td>
<td>0.0004</td>
<td>5.11</td>
</tr>
<tr>
<td>Linear(^a)</td>
<td>0.0259</td>
<td>6.05</td>
</tr>
<tr>
<td>CIM(^a)</td>
<td>0.0525</td>
<td>7.03</td>
</tr>
<tr>
<td>Poly(^a)</td>
<td>0.0125</td>
<td>5.56</td>
</tr>
<tr>
<td>Full-Order CFD(^b)</td>
<td>111</td>
<td>37,000(^c)</td>
</tr>
</tbody>
</table>

\(^a\) One 2.60 GHz Opteron processor core, 2.0 GB RAM.
\(^b\) Thirty-two 2.60 GHz Opteron processor cores, 2.0 GB RAM each.
\(^c\) Projected time.

The “Uncoupled” case does not represent surrogate modeling, it simply uses the initial conditions for the pressure and the “Linear” heat flux model, which is
not a surrogate but a precomputed single value for the heat transfer coefficient. Thus, the 5 hr computational time for 120 seconds of response of this case is primarily due to the thermal model, structural model, and the interface developed in Ref. [128]. Therefore, though the time required to compute one iteration of the aerodynamic loads for the surrogate models are several orders of magnitude more expensive, the total computational cost is minimally increased. The increase in cost of the “Linear” case compared to the “Uncoupled” case is not due to the aerodynamic heating, but due to the more expensive pressure model used in the “Linear” case. As with the HSV panel, the use of CFD within fluid-thermal-structural analysis is impractical even for 2 minutes of response. Finally, note that the computational overhead required to generate the 1500 training snap shots for the “CIM” surrogate is around 2.5 hrs per snap shot on 32 processor cores.

### 8.2 Three-Dimensional Fluid-Thermal Structural Simulation

Next, the computational cost of the 3-D shock impingement panel and the control surface models are discussed. The cost of these models is assessed by extracting the cost of a 2-D thermal and structural analysis and combining that with the cost of the 3-D aerodynamic loads for the 3-D models.
8.2.1 Three-Dimensional Shock Impingement Panel

The cost of 120 seconds of fluid-thermal-structural simulation for the 3-D shock impingement panel using the “Linear”, “Iso”, and “CIM” models as well as a full-order CFD analysis is considered by implementing the underlying thermal, structural, and interface cost from the 2-D shock impingement analysis. The wall time is estimated at 5.1 hrs based on the 2-D “Uncoupled” analysis in Table 8.2. The computational expense of the approaches are compared in Table 8.3. The computational times are based on a fluid-structural time-step of 0.0001 sec and a fluid-thermal time-step of 0.001 sec [128]. The first column represents the computational time required to generate one solution of the pressure and the heat flux. The final column represents the predicted computational time required to produce 120 seconds of fluid-thermal-structural response assuming a 5.1 hr cost for the thermal, structural, and interface.

The cost of one iteration of the aerodynamic heating and pressure for the 3-D “CIM” surrogate is only two-hundredths of a second more expensive than the 2-D “CIM” surrogate. Thus, the predicted cost of 120 seconds of fluid-thermal-structural response increases by less than one hour. However, the cost of the 3-D full-order CFD is more than 5 times as expensive as the 2-D full-order CFD, thus the predicted fluid-thermal-structural response of the 3-D model is over 24 years. Therefore, computational cost reduction for surrogate modeling dramatically increases for three-dimensional problems. Note that the computational overhead re-
Table 8.3: Computational cost of different fluid-thermal-structural modeling approaches for the 3-D panel with shock impingement.

<table>
<thead>
<tr>
<th>Method</th>
<th>1 Iteration of Aero Loads (seconds)</th>
<th>120 seconds of Response (hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear&lt;sup&gt;a&lt;/sup&gt;</td>
<td>0.0311</td>
<td>6.24</td>
</tr>
<tr>
<td>Iso&lt;sup&gt;a&lt;/sup&gt;</td>
<td>0.0467</td>
<td>6.81</td>
</tr>
<tr>
<td>CIM&lt;sup&gt;a&lt;/sup&gt;</td>
<td>0.0786</td>
<td>7.98</td>
</tr>
<tr>
<td>Full-Order CFD&lt;sup&gt;b&lt;/sup&gt;</td>
<td>650</td>
<td>217,000</td>
</tr>
</tbody>
</table>

<sup>a</sup> One 2.60 GHz Opteron processor, 2.0 GB RAM.
<sup>b</sup> Ninety-four 2.60 GHz Opteron processor cores, 2.0 GB RAM each.
<sup>c</sup> Projected time based on 2-D F-T-S simulation.

required to generate the 1500 training snap shots for the “CIM” surrogate is around 4.2 hrs per snap shot on 94 processor cores. Thus, the cost of generating the surrogate also increases for the three-dimensional model.

8.2.2 Control Surface

For the control surface, the cost of fluid-thermal-structural analysis is predicted using the 2-D HSV simulation cost of 43.7 hrs for the thermal model, structural model, interface, and integration. These computational times are based on a time-step size of 0.0005 sec for the fluid-structural (F-S) analysis and 0.1 sec for the fluid-thermal (F-T) analysis [21]. The only surrogate model considered for the control surface was the “Poly” method described in Section 5.3.3. The most accurate cor-
rection for the steady-state surrogate was a second-order piston theory correction, thus computational times are considered for this model (“Poly–PT²”). The computational expense of full-order CFD is also included, and is approximated from the time required for one time step in an unsteady analysis, starting from a converged steady-state solution. The computational expenses of these models are compared in Table 8.4. The first column represents the computational time required to generate one solution of the pressure and the heat flux. The final column represents the computational time required to produce 1500 seconds of fluid-thermal-structural response based on the HSV panel simulation.

Table 8.4: Computational cost of different fluid-thermal-structural modeling approaches for the control surface.

<table>
<thead>
<tr>
<th>Method</th>
<th>1 Iteration of Aero Loads (seconds)</th>
<th>1500 seconds of Response (hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Poly–PT²</td>
<td>0.0882 sec</td>
<td>92.82</td>
</tr>
<tr>
<td>Full-Order CFD</td>
<td>150 sec</td>
<td>83,600</td>
</tr>
</tbody>
</table>

a One 2.60 GHz Opteron processor core, 2.0 GB RAM.
b Twenty-four 2.60 GHz Opteron processor cores, 2.0 GB RAM each.
c Projected time based on 2-D F-T-S simulation

The cost of the 3-D surrogate (“Poly–PT²”) is similar to the cost of the 2-D “CIM” surrogate listed in Table 8.1, thus the projected fluid-thermal-structural response costs are also similar. The full-order CFD model is several orders of magnitude more expense than the surrogate model. Therefore, the fluid-thermal-
structural response using full-order CFD of such a control surface using the time
marching schemes developed in Ref. [21] will require 9.5 years to compute. Fi-
nally, note that the computational overhead required to generate sample points
and train the surrogate is relatively small when parallel processing is used. For
example computation of 1500 sample points on 240 processor cores requires ap-
proximately 13 hours. Furthermore, once the sample points are generated several
flight trajectories can be analyzed simultaneously.
Chapter 9

Concluding Remarks

9.1 Principal Conclusions Obtained in this Study

The fundamental studies conducted in this dissertation on high-speed aerodynamic heating, pressure, and fluid-thermal-structural response allow one to reach several useful conclusions. The conclusions derived from aerodynamic heating studies are presented first, followed by the aerodynamic pressure studies. Finally, the conclusions from the fluid-thermal-structural response studies are presented.

High-Speed Aerodynamic Heating

1. There is up to a 60% variation in aerodynamic heating predictions associated with turbulence modeling in hypersonic flow. Additionally, it is found that boundary layer displacement effects significantly impact chordwise aerodynamic heating profiles on skin panels in hypersonic flow.

2. A quasi-static representation of the aerodynamic heating is found to be acceptable for fluid-thermal-structural analysis. This quasi-static nature of the
aerodynamic heating approximation is exploited in the development of aerodynamic heating surrogates of Computational Fluid Dynamics (CFD) by neglecting unsteady effects in the models.

3. Several novel CFD surrogates are created for the aerodynamic heating. Parameterization of the inputs to the surrogates are separated into trajectory inputs (e.g., Mach number, angle-of-attack, altitude, etc.) and fluid-thermal-structural (F-T-S) inputs. The F-T-S inputs are feedback mechanisms of spatially and temporally varying surface temperatures and deformations.

4. Surrogates are developed using both kriging and Proper Orthogonal Decomposition (POD). The kriging-based models are found to be generally more accurate than POD-based models, while the POD-based models are slightly less expensive.

5. An aerodynamic heating model which varies linearly with respect to surface temperature is developed and found to have average errors up to 40% for both laminar flow over a flat surfaces and turbulent flow over deformed surfaces.

6. A correction for the linear model for variable surface temperatures is developed based on an integration approach. For laminar flow this model reduces the error of the linear approximation by half, however errors are increased relative to turbulent flow.
7. Surrogates based on polynomial surface temperature profiles require bounds on the polynomial coefficients to be arbitrarily determined \textit{a priori}. Extrapolating beyond the bounds on the coefficients led to errors in excess of 50% and inaccurate coupled F-T-S responses.

8. A pointwise approach for coupling surface temperature and aerodynamic heating, which neglects the effect of surface temperature gradients on the heat flux, led to average errors 35% for laminar flow and 10% for turbulent flow.

9. A corrected pointwise model ("CIM") is constructed to account for the effect of surface temperature gradients on the heat flux. This model is found to consistently have the lowest average and maximum errors of all the configurations considered, including 3-D flow fields with deformation and shock impingement. The highest average error for all of the configurations occurred for laminar flow over a flat surface, 6%.

\textbf{High-Speed Aerodynamic Pressure}

1. There is up to a 20% variation in aerodynamic pressure predictions associated with turbulence modeling in hypersonic flow. Additionally, it is found that boundary layer displacement effects and shock impingement significantly impact the unsteady aerodynamic pressures on surfaces in hypersonic flow.

2. Coupling between surface temperature and surface pressure is found to be
negligible, where variations in surface temperature of almost 1000 K led to 3% average differences in the surface pressure.

3. The quasi-steady nature of high speed aerodynamic pressure is exploited by separating the unsteady pressure into steady-state and unsteady components. A comparison of a steady Generalized Aerodynamic Forces (GAFs) with the unsteady GAFs for both 2-D and 3-D flow reveals that the unsteady component of pressure, while generally smaller in magnitude than the steady component, is important for accurately modeling the flow.

4. Simple analytical theories, such as classical piston and local piston theory, provide simple and reasonably accurate approximations for the unsteady component of pressure. The local piston theory approaches provided the best overall approximation for the panels and cases with shock impingement, while second-order piston theory provided the best approach for the control surface.

5. Quasi-static aerodynamic pressure surrogates are constructed from a steady-state CFD database. Unsteady components from classical piston theory and local piston theory are added to the quasi-static pressure surrogates to create unsteady pressure models for F-T-S simulation. These models are found to outperform Euler and classical piston theory for all of the 2-D and 3-D configurations considered.
Fluid-Thermal-Structural Simulation

1. CFD surrogates are implemented into F-T-S analysis of a panel on the inclined surface of a hypersonic vehicle. The use of the surrogates, in place of theoretical aerothermodynamic model approaches for the analysis, can result in almost 100% difference in the onset time to flutter. Furthermore, the predicted post-flutter response is highly sensitive to the aerothermodynamic model approach used.

2. CFD surrogates are also implemented into F-T-S analysis of a compliant panel undergoing forced oscillatory shock impingement. The use of surrogates allows for a systematic analysis of the effect of including specific feedback mechanisms in the F-T-S response, including: variable backpressure, neglecting feedback of deformation, neglecting shock motion, or neglecting motion of any kind. Results show up to an 8% variation in the required backpressure to force the panel to buckle into the flow instead of out.

3. Path dependent responses are observed in the F-T-S analyses. Specifically, the panels experience significant stiffening during post-buckling as a result of the in-plane thermal force, ultimately enabling the panel to sustain higher pressure loading.

4. Linearized aerodynamic heating in the F-T-S analysis results in a conservative estimate of the aerodynamic heat load. This results in reducing the flight
time to panel instabilities by up to 20%.

5. Neglecting either deformation or shock motion for F-T-S simulation can lead to 6% higher average temperatures and 4% larger maximum deflections after 120 seconds. Additionally, oscillating shock impingement led to panel vibrations, particularly evident during the first few seconds of response, when persistent snap through of the panel was observed.

6. There is a slight increase in the computational expense associated with 3-D surrogates compared to 2-D surrogates; while the cost of the full-order CFD increased by at least 5 times from 2-D to 3-D.

7. The developed surrogate approaches provide a reasonably accurate and computationally efficient method for incorporating computational fluid dynamics solutions for the aerothermodynamic loads into a dynamic, long time-record F-T-S analysis. Due to the high computational cost of CFD, these types of models are the only practical means for CFD based F-T-S analysis over long time record hypersonic trajectories.

9.2 Recommendations for Future Research

In this dissertation, a framework has been developed for efficiently constructing surrogates of Computational Fluid Dynamics for fluid-thermal-structural simulations in high speed flows. Emphasis has been placed on including the feedback
mechanisms of spatially and temporally variable surface temperatures and deformations. However, due to the approximations included in the development of the surrogate models (e.g., quasi-static heating, quasi-steady pressure, ideal gas, Reynolds Averaged Navier-Stokes CFD, and pre-determined laminar to turbulent transition locations) there are still significant assessments and developments necessary for the surrogates to be applied for hypersonic vehicle design and analysis.

The error involved in quasi-static heating or quasi-steady pressure are only partially addressed in this dissertation, because little to no full-order data is available for validation. There has been limited study on long duration fluid-thermal-structural simulation, and of these studies the aerothermodynamics are primarily analytical and semi-empirical in nature. This is due to the extreme computational cost of performing full-order fluid-thermal-structural simulation. Furthermore, there is no available experimental data available for validation of the fluid-based loads from fluid-thermal-structural simulations in hypersonic flow. Thus, there is a pressing need to develop both full-order and experimental benchmarks in order to assess the quality of the developed aerothermodynamic surrogate models.

Additionally, the assumptions of: ideal gas, RANS CFD, and pre-determined laminar to turbulent transition for hypersonic flow deserve further investigation. In terms of the ideal gas assumption, a different CFD code which models real gas effects could account for this effect and the surrogates could be re-generated using such a code. There is also a large degree of uncertainty in CFD modeling of hypersonic flow, thus it is not known how poor a RANS approximation may be.
for predicting the mean flow aerothermodynamic loads inside of a fluid-thermal-structural simulation. However, transient fluctuating pressure loads are neglected in a RANS formulation. These loads occur due to turbulent boundary layers, shock impingement, and/or acoustic sources; and these loads may have a significant impact on the fluid-thermal-structural response of a hypersonic vehicle [131]. Therefore in order to use the developed surrogates, modeling of the fluctuating pressure due to these effects should be a priority. Additionally, as shown in this dissertation turbulent heating can be almost 10 times higher than laminar heating, thus prediction of transition from laminar to turbulent flow in the generation of the surrogates should be addressed.

Finally in regards to model reduction, there are a several considerations which warrant further research. Three items that need to be examined are: 1) surrogate technique, 2) sample point identification, and 3) parameterization of deformation for the aerothermodynamic loads. Only two techniques are considered in this dissertation, kriging and POD, however there are a number of other methods available. A worthy objective would be to identify a technique which consistently requires fewer sample points to achieve a desired global accuracy, and which is at a minimum of computational expense relative to the other various methods. In terms of sample point identification, the method employed in this dissertation was a brute force, add more sample points to reduce the error, technique. However, adaptive sampling strategies which aim at reducing the global error of the surrogate and reduce the number of full-order CFD solutions necessary need to be devel-
oped. Finally, deformation is included in the surrogates through the free-vibration modes. This method of parameterizing deformation excludes local deformation (i.e., plasticity). Similar to the aerodynamic heating correction for pointwise surface temperatures, which is capable of handling random temperature profiles, the development of local inclination methods for both aerodynamic heating and the pressure would allow for the inclusion of arbitrary deformations.
Bibliography


10.1 The CFL3D Euler/Navier-Stokes Solver

The NASA Langley CFL3D code [113, 114] is used in this study for CFD solutions to the Euler and Navier-Stokes equations. The CFL3D code uses an implicit, finite-volume algorithm based on upwind-biased spatial differencing to solve either the Euler or the Reynolds-Averaged Navier-Stokes (RANS) equations. Multi-grid and mesh-sequencing are available for convergence acceleration. The algorithm, which is based on a cell-centered scheme, uses upwind-differencing based on either flux-vector splitting or flux-difference splitting, and can sharply capture shock waves. Note that CFL3D is an ideal gas code. Therefore, real gas effects are neglected in the CFD flow analysis.

RANS solvers require turbulence modeling in order to close the RANS equations. However, turbulence modeling remains an active area of research in hypersonic flow and has been shown to be the dominant factor in the accuracy of the solution [45]. Thus, an important consideration for these studies is the uncertainty that exists due to different turbulence models. Note, that no experimental data for the operating conditions considered was available for validation. Thus it is not known which turbulence model is the optimal choice for these studies. Subsequently, several different turbulence models are considered for closure of the RANS equations, including Menter $k - \omega$ SST [47], Wilcox $k - \omega$ [48], and Baldwin-Lomax [49].

10.1.1 CFL3D Fluid Equations

In CFL3D [113], the spatially discretized time dependent Navier-Stokes equations are:

$$\dot{Q}(t) = J R(Q(t))$$  \hspace{1cm} (10.1)

where,

$$Q = [\rho, \rho u, \rho v, \rho w, e]^T$$  \hspace{1cm} (10.2)

$$R = -\frac{\partial (\hat{F} - \hat{F}_v)}{\partial \xi} - \frac{\partial (\hat{G} - \hat{G}_v)}{\partial \eta} - \frac{\partial (\hat{H} - \hat{H}_v)}{\partial \zeta}$$  \hspace{1cm} (10.3)
\[ J = \frac{\partial(\xi, \eta, \zeta)}{\partial(x, y, z)} \]  

(10.4)

The variables \( \hat{F}, \hat{F}_v, \hat{G}, \hat{G}_v, \hat{H}, \) and \( \hat{H}_v \) are the inviscid and viscous flux terms as defined in Ref. [113]. CFL3D solves the nonlinear set of equations using a second order backwards in time temporal discretization about the \( n+1 \) step:

\[ \frac{3Q^{n+1} - 4Q^n + Q^{n-1}}{2J\Delta t} = R(Q^{n+1}) + \mathcal{O} (\Delta t^2) \]  

(10.5)

where \( n \) is the time-stepping increment counter and \( \Delta t \) is the time step. Because \( R \) is a nonlinear function of \( Q \), CFL3D utilizes pseudo time-stepping to perform sub-iterations to converge \( Q^{n+1} \):

\[ \frac{(Q_{m+1} - Q_m)}{J\Delta \tau} + \frac{(1 + \frac{1}{2})(Q_{m+1} - Q^n) - \frac{1}{2}(Q^n - Q^{n-1})}{J\Delta t} = R(Q_{m+1}) \]  

(10.6)

where \( m \) is the pseudo time-stepping sub-iteration counter, and \( \Delta \tau \) is the pseudo time step. The residual vector \( R(Q^{n+1}) \) is approximated using a linearization:

\[ R(Q^{n+1}) \approx R(Q^n) + \frac{\partial R}{\partial Q} \Delta Q \]  

(10.7)

The time stepping becomes second order accurate as the sub-iteration fluxes converge:

\[ \lim_{m \to \infty} Q_{m+1} = Q^{n+1} \]  

(10.8)

### 10.1.2 CFL3D Unsteady Mesh Deformation

Forced unsteady motion in CFL3D can be included through either rigid body motion and/or surface deformation. For this work, only surface deformation is considered. In CFL3D the deformation is included through user specified modal motion, given by [114],

\[ w(x, y, z, t) = \sum_{i=1}^{n} a_i(t) \Phi_i(x, y, z) \]  

(10.9)

where, \( n \) is the number of modes included, \( \Phi_i(x, y, z) \) are the constant in time spatial modeshapes, and \( a_i(t) \) are the time dependent amplitudes. CFL3D offers several options for the amplitudes [114], including harmonic perturbation, Gaussian pulse, and a step pulse. For this work, only the harmonic perturbation is considered, thus:

\[ a_i(t) = A_i \sin(2\pi f_i t) \]  

(10.10)

where, \( A_i \) and \( f_i \) are the amplitude and frequency of the \( i^{th} \) mode, and \( t \) is the time. The fluid mesh deformation is then completed by the Hartwich and Agrawal [117] exponential decay method combined with trans-finite interpolation for the interior mesh points.
10.2 Generic Panel Mesh Convergence

A number of results for this panel are conducted at the conditions of Mach 3.0 and 24 km altitude, thus the mesh convergence is conducted at these conditions. The turbulence model considered for mesh convergence is the Menter $k - \omega$ SST turbulence model, because it is a well established turbulence model for high speed flows [52]. At these conditions the average $y^+$ value at the wall is 0.111. Two coarser meshes of 80,600 cells and 20,150 cells are generated by halving number of grid points in the mesh horizontally and vertically. These meshes are created in order to demonstrate convergence of the lift and drag coefficients. All three meshes are investigated for both laminar and turbulent flow. The surface temperature is set to free stream and deformation is included in the turbulent flow case by setting $a_1 = 25mm$ and $a_2 - a_6 = 0$.

Results are shown in Figs. 10.1 and 10.2 for laminar and turbulent flow, respectively. Figures 10.1(a) and 10.2(a) illustrate the decrease in the residual with iterations, and Figs. 10.1(b,c) and 10.2(b,c) illustrate the convergence of the lift (b) and drag (c) coefficients. Note for laminar flow the lift and drag coefficient are converged to 7.5% and 11.9% between the finer two meshes, while for turbulent flow the coefficients are converged to 0.9% and 0.1%. The reduction in error between the laminar and turbulent results is due primarily to the inclusion of deformation in the turbulent flow case, leading to larger values of the lift and drag coefficients.
Figure 10.1: Residual, lift coefficient, and drag coefficient for the generic panel for laminar flow with 3 mesh densities (20k, 80k, and 320k cells). $M_\infty = 3.0$, $Alt. = 24km$, $T_w = 220K$, $a_1-a_6 = 0$. 
Figure 10.2: Residual, lift coefficient, and drag coefficient for the generic panel for turbulent flow with 3 mesh densities (20k, 80k, and 320k cells). $M_\infty = 3.0$, Alt. = 24km, $T_w = 220K$, $a_1 = 25mm$, $a_2 - a_6 = 0$. 

277
10.3 HSV Panel Mesh Convergence

Most of the results for the HSV panel are conducted with the Menter $k - \omega$ SST turbulence model, again because it is a well established turbulence model for high speed flow [52]. Thus, for mesh convergence of the HSV panel, this turbulence model is chosen. This study is conducted in the middle of the desired parameter space, at $M_2 = 8.4, T_2 = 335K, P_2 = 3691Pa$, where subscript 2 indicates the value is from region 2 behind the oblique shock. The surface temperature is set to 300K and deformation is included by setting $a_1^* = 15.0$ and $a_2^* - a_6^* = 0$. These conditions resulted in an average $y+$ value at the wall of 0.0373. Two coarser meshes of 74,400 cells and 18,600 cells are generated by coarsening the mesh horizontally and vertically for the purpose of demonstrating convergence of the lift and drag coefficients. Results are shown in Fig. 10.3, where Fig. 10.3(a) illustrates the decrease in the residual with iterations, and Figs. (b) and (c) illustrate the convergence of the lift (b) and drag (c) coefficients. Note that between the finer two meshes the coefficients are converged to 1.3% and 2.3% for the lift and drag coefficients, respectively.

Several unsteady Navier-Stokes and Euler CFD simulations are conducted for the HSV panel for verification of the unsteady pressure surrogate models. The time step size and number of sub-iterations for each case are chosen based on convergence of the lift and drag coefficients. An example of the lift and drag coefficients for one of the Navier-Stokes results is shown in Fig. 10.4, and for one of the Euler solutions in Fig. 10.5. The surface motion is a prescribed sinusoidal oscillation of the first mode at $a_1^* = 5$ and a frequency of 140 Hz, similar to values observed in Ref. [21]. The operating conditions in region 2 are: $M_2 = 6.86, T_2 = 300K, P_2 = 2967Pa$, and the surface temperature is set to 900K. Figures 10.4(a,c) and 10.5(a,c) are the lift and drag coefficients over all of the sub-iterations, while (b,d) are a close up of the coefficients over a few of the sub-iterations in order to depict convergence of the coefficients prior to the next time step. This is most clearly seen in Fig. 10.4(d) for the Navier-Stokes simulation, while it can be seen in both (b) and (d) for the Euler simulation, where each step in solution indicates the next time step.

The time step for the unsteady Euler and Navier-Stokes results is based on the frequency. The time step is computed by taking the time for one cycle ($1/f$) and dividing that value by the number of time steps per cycle. The number of time steps per cycle for the Navier-Stokes and Euler cases are listed in Table 10.1, along with the number of sub-iterations and the CFL number.
Figure 10.3: Residual, lift coefficient, and drag coefficient for the HSV panel for three mesh densities (18.6k, 74k, and 298k cells). \( M_2 = 8.4, T_2 = 335K, P_2 = 3691Pa, T_w = 300K, a_1^* = 15.0, a_2^*-a_6^* = 0. \)

Table 10.1: HSV panel unsteady CFD parameters.

<table>
<thead>
<tr>
<th></th>
<th>Navier-Stokes</th>
<th>Euler</th>
</tr>
</thead>
<tbody>
<tr>
<td>Num. of Time Steps per Cycle</td>
<td>1600</td>
<td>1600</td>
</tr>
<tr>
<td>Sub-iterations</td>
<td>150</td>
<td>35</td>
</tr>
<tr>
<td>CFL-( \tau )</td>
<td>5</td>
<td>5</td>
</tr>
</tbody>
</table>
Figure 10.4: Convergence of the unsteady Navier Stokes lift coefficient and drag coefficient for the HSV panel. $M_2 = 6.86, T_2 = 300K, P_2 = 2967Pa, T_w = 900K$, $a_1^* = 5 \sin(2 \pi t 140), a_2^* - a_6^* = 0.$
Figure 10.5: Convergence of the unsteady Euler lift coefficient and drag coefficient for the HSV panel. $M_2 = 6.86$, $T_2 = 300K$, $P_2 = 2967Pa$, $a_1^* = 5 \sin(2 \pi t 140)$, $a_2^*-a_6^* = 0$. 

(a) Lift Coefficient Over All Sub-Iterations. (b) Lift Coefficient Convergence Over A Few Sub-Iterations. 

(c) Drag Coefficient Over All Sub-Iterations. (d) Drag Coefficient Convergence Over A Few Sub-Iterations.
10.4 Two-Dimensional Shock Impingement Panel Mesh

Convergence

The mesh convergence study for the 2-D shock impingement panel is conducted at Mach 3.0, 24 km altitude, with the surface temperature set to 220K, the shock-generator wedge angle at $10^\circ$, and a deformation of $a_1^* = 20.0$ and $a_2^*-a_6^* = 0$. The turbulence model chosen is the Menter $k-\omega$ SST turbulence model, again because it is a well established turbulence model for high speed flow [52]. At these conditions, the average $y+$ value at the wall is 0.0851. Two coarser meshes of 147,000 cells and 36,700 cells are generated by coarsening the mesh horizontally and vertically, in order to demonstrate convergence of the lift and drag coefficients. Results are shown in Fig. 10.6. Figure 10.6(a) shows the decrease in the residual with iterations, and Figs. 10.6(b,c) show the convergence of the lift (b) and drag (c) coefficients. Note that between the finer two meshes the coefficients are converged to 45.9% and 0.3% for the lift and drag coefficients, respectively. Note, the coefficient of lift is small and therefore large errors exist between the meshes, the drag coefficient is much larger, and the convergence is excellent at 0.3%.

Several unsteady Navier-Stokes and Euler CFD simulations are conducted for the 2-D shock impingement panel, including the effect of a stationary versus a moving shock generator. The time step size and number of sub-iterations for each case are chosen based on convergence of the lift and drag coefficients. An example of the lift and drag coefficients for one of the stationary shock generator Navier-Stokes results is shown in Fig. 10.7, and for one of the Euler solutions in Fig. 10.8 for a single cycle of the panel oscillation. The free stream flow conditions are listed in Table 2.5. Deformation is prescribed through a sinusoidal oscillation of the first mode at $a_1^* = 5$ at its fundamental frequency of 61.01 Hz. The surface temperature is set to 440 K, and the stationary shock generator angle is set to $10^\circ$.

An example of the lift and drag coefficients for one of the moving shock generator Navier-Stokes results is shown in Fig. 10.9, and for one of the Euler solutions in Fig. 10.10. For these cases the shock generator oscillates at 10 Hz $\pm 3^\circ$ about a nominal $10^\circ$, the surface deformation is for the first mode oscillating at $a_1^* = 5$ at 50 Hz, and the surface temperature is set again to 440 K. The results shown are for five panel cycles corresponding to one shock generator cycle. Figures 10.7 to 10.10 (a) and (c) are the lift and drag coefficients over all of the sub-iterations, while (b) and (d) are a close up of the coefficients over a few of the sub-iterations in order to depict convergence of the coefficients prior to the next time step.

Several frequencies are considered for this panel, thus the time step for the unsteady Euler and Navier-Stokes results is based on the frequency. The time step is computed by taking the time for one cycle ($1/f$) and dividing that value by the
Figure 10.6: Residual, lift coefficient, and drag coefficient for the 2-D shock impingement panel for three mesh densities (36.7k, 147k, and 588k cells). $M_\infty = 3.0$, $Alt. = 24.0km$, $T_w = 220K$. Shock generator: $= 10^\circ$. Panel: $a_1^* = 20.0$, $a_2^*-a_6^* = 0$. 

283
Figure 10.7: Convergence of the unsteady Navier Stokes lift coefficient and drag coefficient for the 2-D shock impingement panel with the stationary shock generator at 10°. $M_\infty = 3.0$, Alt. = 24 km, $T_w = 440 K$. Panel: $a_1^* = 5 \sin(2 \pi t 61.01)$, $a_2^*-a_6^* = 0$. 

(a) Lift Coefficient Over All Sub-Iterations. 
(b) Lift Coefficient Convergence Over A Few Sub-Iterations. 
(c) Drag Coefficient Over All Sub-Iterations. 
(d) Drag Coefficient Convergence Over A Few Sub-Iterations.
Figure 10.8: Convergence of the unsteady Euler lift coefficient and drag coefficient for the 2-D shock impingement panel with the stationary shock generator at 10°. $M_\infty = 3.0$, Alt. = 24km. Panel: $a_1^* = 5\sin(2\pi t 61.01)$, $a_2^* - a_6^* = 0$. 

(a) Lift Coefficient Over All Sub-Iterations. 
(b) Lift Coefficient Convergence Over A Few Sub-Iterations. 
(c) Drag Coefficient Over All Sub-Iterations. 
(d) Drag Coefficient Convergence Over A Few Sub-Iterations.
Figure 10.9: Convergence of the unsteady Navier Stokes lift coefficient and drag coefficient for the 2-D shock impingement panel with an oscillating shock generator angle: 10 Hz ± 3° about a nominal 10°. $M_\infty = 3.0$, Alt.= 24km, $T_w = 440K$. Panel: $a_1^* = 5\sin(2 \pi t 50)$, $a_2^* - a_6^* = 0$. 

Panel 1:
- (a) Lift Coefficient Over All Sub-Iterations.
- (b) Lift Coefficient Convergence Over A Few Sub-Iterations.

Panel 2:
- (c) Drag Coefficient Over All Sub-Iterations.
- (d) Drag Coefficient Convergence Over A Few Sub-Iterations.
Figure 10.10: Convergence of the unsteady Euler lift coefficient and drag coefficient for the 2-D shock impingement panel with an oscillating shock generator angle: $10 \, \text{Hz} \pm 3^\circ$ about a nominal $10^\circ$. $M_\infty = 3.0$, Alt. = $24\, \text{km}$, $T_w = 440\, \text{K}$. Panel: $a_1^* = 5\sin(2\pi t \, 50)$, $a_2^*-a_6^* = 0$. 

(a) Lift Coefficient Over All Sub-Iterations. 
(b) Lift Coefficient Convergence Over A Few Sub-Iterations. 
(c) Drag Coefficient Over All Sub-Iterations. 
(d) Drag Coefficient Convergence Over A Few Sub-Iterations.
number of time steps per cycle. The number of time steps per cycle for the Navier-
Stokes and Euler cases are listed in Table 10.2 for the stationary shock cases, and
in Table 10.3 for the moving shock cases, along with the number of sub-iterations
and the CFL number.

Table 10.2: 2-D shock impingement panel unsteady CFD parameters for the sta-
tionary shock generator cases.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Num. of Time Steps per Cycle</td>
<td>2000</td>
</tr>
<tr>
<td>Sub-iterations</td>
<td>100</td>
</tr>
<tr>
<td>CFL-$\tau$</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Table 10.3: 2-D shock impingement panel unsteady CFD parameters for the oscil-
lating shock generator cases

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Num. of Time Steps per Cycle</td>
<td>2000</td>
</tr>
<tr>
<td>Sub-iterations</td>
<td>200</td>
</tr>
<tr>
<td>CFL-$\tau$</td>
<td>1.0</td>
</tr>
</tbody>
</table>
10.5 Three-Dimensional Shock Impingement Panel Mesh

Convergence

The mesh convergence study for the shock impingement panel is conducted at the same conditions as the 2-D shock impingement panel, Mach 3.0 and 24 km altitude. The surface temperature set to 220K, the shock-generator wedge angle is set at $10^\circ$, and the deformation is set to $a_1^* = 20.0$ and $a_2^* - a_6^* = 0$. At these conditions the average $y+$ value at the wall is 0.0812. One coarser mesh of 930,000 cells and one finer mesh of 59.52 million cells are generated, in order to demonstrate convergence of the lift and drag coefficients. Results are shown in Fig. 10.11. Figure 10.11(a) illustrates the decrease in the residual with iterations, and Figs. 10.11(b,c) illustrate the convergence of the lift (b) and drag (c) coefficients. Note that between the finer two meshes the coefficients are converged to 8.1% and 3.9% for the lift and drag coefficients, respectively. Note, the lift coefficient is small for this wind tunnel model, thus changes in grid density result in larger errors.

For time step and sub-iteration convergence, the 3-D shock impingement panel is deformed in the first mode from -5 to +5 times the thickness at a frequency of 50 Hz, while the shock generator is oscillated according to Eq. (2.5) at a frequency of 10 Hz. The flow conditions for this analysis are the Mach 2.0 flow conditions from Table 2.7, and the surface temperature is set equal to the free stream temperature. The time step is based on the number of time steps required per cycle to converge the solution at these frequencies, and the sub-iterations are chosen based on convergence of the lift and drag coefficients for this result. The time step is computed by taking the time for one cycle ($1/f$) and dividing that value by the number of time steps per cycle. The number of time steps per cycle for the Navier-Stokes and Euler cases are listed in Table 10.3 for the moving shock cases, along with the number of sub-iterations and the CFL number. The Navier-Stokes convergence results are shown in Fig. 10.12, and the Euler results in Fig. 10.13. Figures 10.12 and 10.13 (a) and (c) are the lift and drag coefficients over all of the sub-iterations and time steps, while (b) and (d) are a close up of the coefficients over a few of the sub-iterations in order to depict convergence of the coefficients prior to the next time step.
Figure 10.11: Residual, lift coefficient, and drag coefficient for the 3-D shock impingement panel for three mesh densities (930k, 7.4mil, and 59.5mil cells). $M_{\infty} = 3.0$, $Alt. = 24.0$ km, $T_w = 220K$. Shock generator: $10^\circ$. Panel: $a_1^* = 20.0$, $a_2^*-a_6^* = 0$.

Table 10.4: 3-D shock impingement panel unsteady CFD parameters.

<table>
<thead>
<tr>
<th></th>
<th>Navier-Stokes</th>
<th>Euler</th>
</tr>
</thead>
<tbody>
<tr>
<td>Num. of Time Steps per Cycle</td>
<td>2000</td>
<td>1000</td>
</tr>
<tr>
<td>Sub-iterations</td>
<td>150</td>
<td>70</td>
</tr>
<tr>
<td>CFL-$\tau$</td>
<td>1.0</td>
<td>1.0</td>
</tr>
</tbody>
</table>
Figure 10.12: Convergence of the unsteady Navier Stokes lift coefficient and drag coefficient for the 3-D shock impingement panel with an oscillating shock generator angle: $10 \text{ Hz} \pm 3^\circ$ about a nominal $10^\circ$. $M_\infty = 2.0$, $T_\infty = 215.5K$, $P_\infty = 44093Pa$, $T_w = 215.5K$. Panel: $a_1^* = 5\sin(2 \pi t 50)$, $a_2^*-a_6^* = 0$. 
Figure 10.13: Convergence of the unsteady Euler lift coefficient and drag coefficient for the 3-D shock impingement panel with an oscillating shock generator angle: $10 \ Hz \pm 3^\circ$ about a nominal $10^\circ$. $M_\infty = 2.0$, $T_\infty = 215.5K$, $P_\infty = 44093Pa$, $T_w = 215.5K$. Panel: $a_1^* = 5sin(2 \pi \ t \ 50)$, $a_2^* - a_6^* = 0$. 

(a) Lift Coefficient Over All Sub-Iterations.

(b) Lift Coefficient Convergence Over A Few Sub-Iterations.

(c) Drag Coefficient Over All Sub-Iterations.

(d) Drag Coefficient Convergence Over A Few Sub-Iterations.
10.6 Control Surface Mesh Convergence

The mesh convergence study for the control surface is conducted at Mach 8.0, 40.0 km altitude, angle-of-attack is 4.0°, side-slip angle is 0°. The surface temperature set to 1200 K, and no surface deformation is included. The turbulence model implemented is the Baldwin-Lomax turbulence model, which is consistent with all of the results using this model. At these conditions the average y+ value at the wall is 0.564. One coarser mesh of 34,000 cells and one finer mesh of 2.2 million cells are generated, in order to demonstrate convergence of the lift and drag coefficients. Results are shown in Fig. 10.14. Figure 10.14(a) illustrates the decrease in the residual with iterations, and Figs. 10.11(b,c) and illustrate the convergence of the lift (b) and drag (c) coefficients. Note that between the finer two meshes the coefficients are converged to 1.3% and 29.8% for the lift and drag coefficients, respectively. Note, this model is primarily used for pressure studies, thus the convergence of the coefficient of lift is used to determine the required mesh density.

For time step and sub-iteration convergence, the control surface is deformed in the first mode from -10% to +10% tip deflection at a frequency of 200 Hz, which is the highest frequency considered in this study. The Mach number is set to 8.0 with 40.0 km altitude, the angle-of-attack and side-slip angle are set to 0°, and a surface temperature of 1200 K. The number of time steps per cycle and the number of sub-iterations are chosen based on convergence of the lift and drag coefficients for this result. The Navier-Stokes result is shown in Fig. 10.15, and the Euler solution in Fig. 10.16. Figures 10.15 and 10.16 (a) and (c) are the lift and drag coefficients over all of the sub-iterations and time steps, while (b) and (d) are a close up of the coefficients over a few of the sub-iterations in order to depict convergence of the coefficients prior to the next time step. The number of time steps per cycle for the Navier-Stokes and Euler cases are listed in Table 10.5, along with the number of sub-iterations and the CFL number. Finally, note that all unsteady cases for the control surface are performed on the finest (2.2 million cell) grid, due to a lack of time step convergence with the 270,000 cell mesh.

Table 10.5: Control surface unsteady CFD parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Num. of Time Steps per Cycle</td>
<td>800</td>
</tr>
<tr>
<td>Sub-iterations</td>
<td>100</td>
</tr>
<tr>
<td>CFL-τ</td>
<td>10.0</td>
</tr>
<tr>
<td>CFD Mesh (# of Cells)</td>
<td>2.2 million</td>
</tr>
</tbody>
</table>
Figure 10.14: Residual, lift coefficient, and drag coefficient for the control surface for three mesh densities (34k, 270k, and 2.2mil cells). $M_\infty = 8.0$, Alt. = 40.0 km, $T_w = 1200K$, $a_1^* = 0$, $a_2^* = 0$. 

---

(a) Residual.

(b) Lift Coefficient.

(c) Drag Coefficient.
Figure 10.15: Convergence of the unsteady Navier Stokes lift coefficient and drag coefficient for the control surface. $M_\infty = 8.0$, $Alt. = 40.0$ km, $T_w = 1200K$, $a_1^* = 1.0\sin(2 \pi t 200)$, $a_2^* = 0$. 

(a) Lift Coefficient Over All Sub-Iterations. 
(b) Lift Coefficient Convergence Over A Few Sub-Iterations. 
(c) Drag Coefficient Over All Sub-Iterations. 
(d) Drag Coefficient Convergence Over A Few Sub-Iterations. 

$M_\infty = 8.0$, $Alt. = 40.0$ km, $T_w = 1200K$, $a_1^* = 1.0\sin(2 \pi t 200)$, $a_2^* = 0$. 

295
Figure 10.16: Convergence of the unsteady Euler lift coefficient and drag coefficient for the control surface. $M_\infty = 8.0$, $Alt. = 40.0 \text{ km}$, $a_1^* = 1.0 \sin(2 \pi t 200)$, $a_2^* = 0$. 
10.7 Eckert’s Reference Methods

Eckert’s reference enthalpy methods have been used extensively in approximate analyses to efficiently model convective heating of high-speed aerospace vehicles [74–78, 81]. This method uses boundary layer relations from incompressible flow theory with flow properties evaluated at a reference condition to account for the effects of compressibility.

Eckert’s reference enthalpy is given by Eq. (10.11). The adiabatic wall enthalpy, the total enthalpy, and the recovery factor for turbulent flow are given in Eqs. (10.12) – (10.14), respectively [37]. Note that local velocity and enthalpy at the edge of the boundary layer are determined from the local edge Mach number and temperature, respectively. Mach number and temperature at the edge of the boundary layer are computed using the local pressure from an inviscid aerodynamic theory. Following Culler and McNamara [21], piston theory is used in this study.

\[
H_e = H_e + 0.50(H_w - H_e) + 0.22(H_{AW} - H_e) \quad (10.11)
\]

\[
H_{AW} = r(H_0 - H_e) + H_e \quad (10.12)
\]

\[
H_0 = H_e + \frac{U_e^2}{2} \quad (10.13)
\]

\[
r = (Pr_*)^{1/3} \quad (10.14)
\]

Using flow properties evaluated at the reference enthalpy, the aerodynamic heat flux is computed using Eq. (10.15). The Stanton number is determined from the Colburn-Reynolds analogy [37, 132] provided in Eq. (10.16), and the local skin friction coefficient is computed using the Schultz-Grunow formula [37, 132] given in Eq. (10.17). The local Reynolds number is defined in Eq. (10.18) and is computed using the distance from the onset of transition from laminar to turbulent flow to the point of interest [132]. Turbulent flow is assumed over the surface.

\[
Q(x) = C_{H*}\rho_*U_e (H_{AW} - H_w) \quad (10.15)
\]

\[
C_{H*} = \frac{c_{f*}}{2} \left(\frac{1}{Pr_*}\right)^{2/3} \quad (10.16)
\]

\[
c_{f*} = \frac{0.370}{(\log_{10} \text{Re}_{xx})^{2.584}} \quad (10.17)
\]

\[
\text{Re}_{xx} = \frac{\rho_*U_e x}{\mu_*} \quad (10.18)
\]

297
In addition to Eqs. (10.11) – (10.18), temperature-enthalpy relations are needed to determine the values of enthalpy at the wall and at the edge of the boundary layer. These relations are also required to determine a temperature from the reference enthalpy in order to evaluate reference density and reference viscosity using the ideal gas law and Sutherland’s law, respectively [72]. If the flow is assumed to be calorically perfect, i.e., constant specific heat, then Eq. (10.19) is used and the reference enthalpy method is equivalent to the reference temperature method [37]. However, since this study is concerned with hypersonic flow, in which the specific heat may vary through the boundary layer due to high temperatures and real gas effects [72], temperature-enthalpy tables [133] that include the effect of dissociation based on equilibrium air properties are employed.

\[
H = c_p T
\]  
(10.19)
10.8 Chapman and Rubesin’s Method

The following sections detail the 2-D compressible laminar aerodynamic heating model developed by Chapman and Rubesin [67].

10.8.1 Laminar Boundary Layer Equations

Beginning with the continuity, momentum, and energy equations for a compressible laminar boundary layer over a flat plate, assuming constant values of $c_p$ and $Pr$ [67]:

\[
\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} = 0 \tag{10.20}
\]

\[
\rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} = \frac{\partial}{\partial y} \left( \mu \frac{\partial u}{\partial y} \right) \tag{10.21}
\]

\[
\rho u \frac{\partial T}{\partial x} + \rho v \frac{\partial T}{\partial y} = \frac{1}{Pr} \frac{\partial}{\partial y} \left( \mu \frac{\partial T}{\partial y} \right)^2 \tag{10.22}
\]

Note, $y$ in these equations refers to the transverse direction and $x$ is the usual flow wise direction. The variables are then changed from $x, y$ to $x, \Psi$. Where the stream function $\Psi$ is defined by,

\[
u = \frac{\rho_\infty}{\rho} \frac{\partial \Psi}{\partial y} \tag{10.23}
\]

and

\[
v = -\frac{\rho_\infty}{\rho} \frac{\partial \Psi}{\partial x} \tag{10.24}
\]

Using the stream function, $\Psi$, the continuity equation is automatically satisfied, and the momentum and energy equations become [67],

\[
\frac{\partial u}{\partial x} = \frac{\mu_\infty}{\rho_\infty} \frac{\partial}{\partial \Psi} \left( C_u \frac{\partial u}{\partial \Psi} \right) \tag{10.25}
\]

\[
\frac{\partial T}{\partial x} = \frac{\mu_\infty}{Pr \rho_\infty} \frac{\partial}{\partial \Psi} \left( C_u \frac{\partial T}{\partial \Psi} \right) + \frac{\mu_\infty C}{c_p \rho_\infty} \left( \frac{\partial u}{\partial \Psi} \right)^2 \tag{10.26}
\]

where,

\[
C = \sqrt{\frac{T_w}{T_\infty} \left( \frac{T_\infty + S}{T_w + S} \right)} \tag{10.27}
\]
is the Chapman-Rubesin constant. The parameter $C$ assumes a linear relationship between viscosity ratio and the temperature ratio:

$$\frac{\mu}{\mu_\infty} = C \frac{T}{T_\infty}$$ (10.28)

However, note that Chapman and Rubesin [67] use a constant for the wall temperature in Eq. (10.27), $T_w$, which is the average wall temperature. The parameter $C$ with the constant $T_w$ is introduced in order to simplify the solution of the compressible boundary layer equations.

Equations (10.25) and (10.26) are further simplified and decoupled by implementing the following dimensionless variables [67],

$$u^* = \frac{u}{U_\infty}, \quad x^* = \frac{x}{L}, \quad T^* = \frac{T}{T_\infty}, \quad \rho^* = \frac{\rho}{\rho_\infty}, \quad \mu^* = \frac{\mu}{\mu_\infty}, \quad \Psi^* = \frac{\Psi}{\sqrt{\nu_\infty U_\infty L}C}$$ (10.29)

Using these variables, Eqs. (10.25) and (10.26) become,

$$\frac{\partial u^*}{\partial x^*} = \frac{\partial}{\partial \Psi^*} \left( u^* \frac{\partial u^*}{\partial \Psi^*} \right)$$ (10.30)

$$\frac{\partial T^*}{\partial x^*} = \frac{1}{Pr} \frac{\partial}{\partial \Psi^*} \left( u^* \frac{\partial T^*}{\partial \Psi^*} \right) + (\gamma - 1) M_\infty^2 u^* \left( \frac{\partial u^*}{\partial \Psi^*} \right)^2$$ (10.31)

with boundary conditions,

$$u^*(x^*, 0) = 0, \quad u^*(x^*, \infty) = 1$$

$$T^*(x^*, 0) = T_w, \quad T^*(x^*, \infty) = 1$$ (10.32)

Equation (10.30) does not depend on $\rho^*$, thus the solution for $u^*$ is the classical incompressible Blasius solution [67],

$$u^* = \frac{1}{2} f'(\eta)$$ (10.33)

where,

$$f(\eta) = \frac{\Psi^*}{\sqrt{x^*}}, \quad f'(0) = f(0) = 0, \quad f'(\infty) = 2$$ (10.34)

Substituting the Blasius solution for $u^*$, Eq. (10.33), into Eq. (10.31) results in,

$$\frac{\partial T^*}{\partial x^*} = \frac{1}{2Pr} \frac{\partial}{\partial \Psi^*} \left( f'(\eta) \frac{\partial T^*}{\partial \Psi^*} \right) + \frac{\gamma - 1}{8} M_\infty^2 \left( f''(\eta) \right)^2 \frac{u^*}{x^*} \frac{1}{f'(\eta)}$$ (10.35)
Equation (10.35) is then transformed from variables \((x^*, \Psi^*)\) to \((x^*, \eta)\) by,

\[
\left( \frac{\partial}{\partial x^*} \right)_\Psi = \left( \frac{\partial}{\partial x^*} \right)_\eta - \frac{f}{2x^*f'} \frac{\partial}{\partial \eta}.
\]

\[
\left( \frac{\partial}{\partial \Psi^*} \right)_x = \frac{1}{\sqrt{x^*f'}} \frac{\partial}{\partial \eta}.
\]

The final form for the energy equation is [67],

\[
\frac{\partial^2 T^*}{\partial \eta^2} + Pr f(\eta) \frac{\partial T^*}{\partial \eta} - 2Pr f'(\eta)x^* \frac{\partial T^*}{\partial x^*} = -\frac{Pr}{4} (\gamma - 1) M_{\infty}^2 (f''(\eta))^2
\]

(10.37)

with boundary conditions

\[
T^*(x^*, 0) = T_{AW}^* + t(x^*), \quad T^*(x^*, \infty) = 1
\]

(10.38)

### 10.8.2 Particular Solution

The particular solution of the inhomogeneous equation can be found by substituting:

\[
T^*(x^*, \eta) = X(x^*) + N(\eta)
\]

into Eq. (10.37), which gives,

\[
\frac{1}{2Pr f'(\eta)} \left( N''(\eta) + Pr f(\eta)N'(\eta) + \frac{Pr}{4} (\gamma - 1) M_{\infty}^2 (f''(\eta))^2 \right) = x^* X'
\]

(10.40)

Both sides of the equation must be equal to a constant, which must be 0 in order to prevent a singularity at \(x = 0\) [67]. Thus,

\[
N''(\eta) + Pr f(\eta)N'(\eta) = -\frac{Pr}{4} (\gamma - 1) M_{\infty}^2 (f''(\eta))^2
\]

(10.41)

The solution to this ordinary differential equation is,

\[
T^*(x^*, \eta) = N(\eta) = 1 + r(\eta) \frac{\gamma - 1}{2} M_{\infty}^2
\]

(10.42)

\[
r(\eta) = \frac{Pr}{2} \int_\eta^\infty (f''(\xi))^P r \int_0^\xi (f''(\theta))^2 - Pr \, d\theta d\xi
\]

where at the wall,

\[
r(0) = 0.845, \quad N(0) = T_{AW}^* = 1 + r(0) \frac{\gamma - 1}{2} M_{\infty}^2
\]

(10.43)

Note, \(r(0)\) is the exactly recovery factor for the adiabatic wall temperature of a laminar flow [72].
10.8.3 Homogenous Solution

The solution of the homogenous equation is obtained by separation of variables,

\[ T^*(x^*, \eta) = X(x^*)Y(\eta) \]  \hfill (10.44)

which yields,

\[
\left( \frac{1}{f'(\eta)} \right) \left( Y''(\eta) + Prf(\eta)Y'(\eta) \right) = 2Prx^* \frac{X'(x^*)}{X(x^*)} \]  \hfill (10.45)

Again, both sides of the equation must equal a constant. Following Ref. [67], the constant is set equal to \( 2 Prn \), which yields:

\[ X_n(x^*) = x^{*n} \]  \hfill (10.46)

and

\[ Y_n''(\eta) + Prf(\eta)Y_n'(\eta) - 2nPrf'(\eta)Y_n(\eta) = 0 \]  \hfill (10.47)

with boundary conditions,

\[ Y_n(0) = 1, \quad Y_n(\infty) = 0 \]  \hfill (10.48)

The general solution to the linear homogenous equation is,

\[ T^*(x^*, \eta) = \sum_{n=0}^{\infty} b_n x^{*n} Y_n(\eta) \]  \hfill (10.49)

where \( b_n \) are constants determined from the boundary conditions, which are:

\[ T^*(x^*, 0) = \sum_{n=0}^{\infty} b_n x^{*n}, \quad T^*(x^*, \infty) = 0 \]  \hfill (10.50)

10.8.4 Complete Solution

The complete solution of Eq. (10.37) is then

\[ T^*(x^*, \eta) = N(\eta) + \sum_{n=0}^{\infty} b_n x^{*n} Y_n(\eta) \]  \hfill (10.51)

with boundary conditions,

\[ T^*(x^*, 0) \equiv T^*_w = T^*_AW + \sum_{n=0}^{\infty} b_n x^{*n} \]  \hfill (10.52)

\[ T^*(x^*, \infty) = 1 \]
Thus, the coefficients $b_n$ can be determined from the infinite series representation of the wall temperature minus the adiabatic wall temperature,

$$T_w^* - T_{AW}^* = \sum_{n=0}^{\infty} b_n x^n$$  \hspace{1cm} (10.53)

### 10.8.5 Heat Flux

Substituting Eq. (10.51) into Eq. (4.1) yields,

$$Q(x) = k_w T_\infty \left( \frac{\partial \eta}{\partial y} \right)_w \sum_{n=0}^{\infty} b_n x^n Y_n'(0)$$ \hspace{1cm} (10.54)

where the values $Y_n'(0)$ from Ref. [67] are shown in Table 4.2. Equation (10.54) can be written as,

$$Q(x) = \frac{k_\infty T_\infty}{2} C_w \sqrt{\frac{\rho_\infty U_\infty}{\mu_\infty x C}} \sum_{n=0}^{\infty} b_n x^n Y_n'(0)$$ \hspace{1cm} (10.55)

The variable $C_w$ is dependent on $T_w$ and therefore also dependent on $x$. $C_w$ is defined by,

$$C_w = \sqrt{\frac{T_w}{T_\infty}} \left( \frac{T_\infty + S}{T_w + S} \right)$$ \hspace{1cm} (10.56)

Note Eq. (10.56) should not be confused with the Chapman and Rubesin constant $C$, which is dependent on the average surface temperature.