ABSTRACT

LOOSELY COUPLED HYPersonic Airflow Simulation over a Thermally Deforming Panel with Applications for a POD Reduced Order Model

by Joshua Gabriel Smith

Predicting surface panel buckling caused by viscous heating around hypersonic vehicles is a complex problem as the interaction between the airflow and the structure results in a physically coupled system that is computationally expensive to solve with traditional computational fluid dynamics (CFD). Reduced order modeling (ROM) via proper orthogonal decomposition (POD) offers improved efficiency in simulating this system over traditional CFD, but must be formed from an existing data set. This study used loosely coupled fluid-thermal-structural simulations to examine Mach 4.24 flow over a deforming stainless steel panel. The methodology and domain from a previous study were used and expanded. ANSYS Fluent CFD software was used to solve the flow field for steady state and transient scenarios. Abaqus finite element analysis (FEA) software was used to examine the panel’s response to applied temperature profiles. Results were compared to the previous study and oblique shock theory for validation. It was found that altering the interval of data transfer between systems could be optimized to balance accuracy and efficiency. Applications associated with creating a POD ROM from the simulation data were also considered.
LOOSELY COUPLED HYPersonic AIRFLOW SIMULATION OVER A THERmALLY DEFORMING PANEL WITH APPLICATIONS FOR A POD REDuced ORDER MODEL

Thesis Report

Submitted to the
Faculty of Miami University
in partial fulfillment of
the requirements for the degree of
Master of Science in Mechanical Engineering

by
Joshua Gabriel Smith
Miami University
Oxford, Ohio
2017

Advisor: Dr. Edgar Caraballo
Reader: Dr. Amit Shukla
Reader: Dr. Andrew Sommers

©2017 Joshua Gabriel Smith
This Thesis Report titled

LOOSELY COUPLED HYPersonic AIRFLOW SIMULATION OVER A THERmALLY DEFORMING PANEL WITH APPLICATIONS FOR A POD REDUCED ORDER MODEL

by

Joshua Gabriel Smith

has been approved for publication by

The College of Engineering and Computing

and

Department of Mechanical and Manufacturing Engineering

______________________________________________________
Dr. Edgar Caraballo

______________________________________________________
Dr. Amit Shukla

______________________________________________________
Dr. Andrew Sommers
# Table of Contents

ABSTRACT ................................................................................................................................. iii

List of Tables ................................................................................................................................ v

List of Figures ............................................................................................................................... vi

Nomenclature .............................................................................................................................. ix

Dedication ...................................................................................................................................... xi

Acknowledgements ....................................................................................................................... xii

Chapter 1: Literature Review ......................................................................................................... 1

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

1.2 Background ........................................................................................................................... 2

1.3 Case Study: Thornton and Dechaumphai (1988) ................................................................... 4

1.4 Hypersonic Fluid Dynamics .................................................................................................. 5

1.4.1. ANSYS Fluent Theory .................................................................................................... 9

1.4.2. Supersonic Oblique Shock Equations ........................................................................... 14

1.5 Panel Deformation and Buckling ....................................................................................... 19

1.5.1. Elastic Deformation ....................................................................................................... 19

1.5.2. Buckling ....................................................................................................................... 20

1.5.3. Abaqus Theory ............................................................................................................. 22

1.6 Proper Orthogonal Decomposition ..................................................................................... 22

Chapter 2: Research Methodology ................................................................................................. 24

2.1 Domain and Mesh Creation ................................................................................................. 24

2.2 MATLAB Model .................................................................................................................. 29

2.3 Steady State Simulations ..................................................................................................... 31

2.4 Structural Simulations ......................................................................................................... 33

2.5 Loosely Coupled Simulations ............................................................................................ 37

2.6 Sinusoidally Driven Simulations ....................................................................................... 38

Chapter 3: Results and Discussion .............................................................................................. 39

3.1 Steady State Simulation Results .......................................................................................... 39

3.1.1. Comparing Turbulence Models .................................................................................. 39

3.1.2. Comparing Velocities ............................................................................................... 43
List of Tables

**Table 1:** Boundary conditions for mesh A. .................................................................................. 25
**Table 2:** Boundary conditions for mesh B. .................................................................................. 27
**Table 3:** Temperature dependent properties of air. ................................................................. 29
**Table 4:** Inputs and outputs of the MATLAB model. .................................................................. 30
**Table 5:** The effect of eccentricity on panel buckling for pinned conditions. ......................... 36
**Table 6:** Thornton and Abaqus panel deflections for various average temperatures through time. ................................................................................................................................. 49
**Table 7:** Thornton and Abaqus average top panel stresses for various average temperatures through time. ................................................................................................................................. 50
**Table 8:** Buckling forces and temperatures from Abaqus FEA for pinned and fixed support boundary conditions. ................................................................................................................................. 51
**Table 9:** Temperature changes to cause buckling from Abaqus FEA and Euler buckling theory. ...................................................................................................................................................... 51
**Table 10:** Temperature averages for the top panel surface at time intervals of $t = 0, 10, 20,$ and $30 \text{s}$ for Thornton and ANSYS Fluent laminar solver with convex boundary conditions. ............ 54
**Table 11:** Heat flux averages for the top panel surface at time intervals of $t = 0, 10, 20,$ and $30 \text{s}$ for Thornton and ANSYS Fluent laminar solver with convex boundary conditions. .......... 55
**Table 12:** Deflection maximums for the top panel surface at time intervals of $t = 0, 10, 20,$ and $30 \text{ seconds}$ for Thornton and ANSYS Fluent laminar solver with convex boundary conditions. 55
**Table 13:** The deflection, average temperature, and average heat flux at $t = 10 \text{s}$ for loosely coupled laminar simulations with transfer intervals of $10 \text{s}, 5 \text{s}, 2.5 \text{s},$ and $1 \text{s}$ .............. 58
**Table 14:** The deflection, average temperature, and average heat flux at $t = 2.5 \text{s}$ for laminar, $k - \epsilon,$ and $k - \omega$ model simulations. ......................................................................................................................... 60
**Table 15:** Constant properties of AM-350 stainless steel from High Temp Metals (2015). ....... 89
**Table 16:** Temperature dependent properties of AM-350 stainless steel from High Temp Metals (2015). ................................................................................................................................. 90
**Table 17:** Temperature dependent specific heat properties of air (Cengel & Boles, 2002). ....... 90
List of Figures

Figure 1: Diagram of a normal shock, oblique shock, and expansion wave. .............................. 14
Figure 2: Mesh A schematic with boundary conditions and dimensions. ................................. 25
Figure 3: Mesh B schematic with boundary conditions and dimensions. ................................. 27
Figure 4: Plots of mesh convergence for flow on mesh B after 5 s for (A) boundary layer thickness
and (B) average top panel surface temperature ................................................................. 28
Figure 5: Sample output from MATLAB model showing shock waves and expansion waves over
a trapezoidal geometry ....................................................................................................... 31
Figure 6: The five geometric setups for steady state simulations .............................................. 32
Figure 7: Boundary conditions for elastic deformation structural analysis. .............................. 33
Figure 8: Boundary conditions for structural buckling analysis .............................................. 34
Figure 9: Flow diagram of loosely coupled fluid-thermal-structural simulation procedure ..... 38
Figure 10: Plot of absolute pressure vs position for MATLAB model and different ANSYS Fluent
models on the convex trapezoidal geometry at $M = 4.24$. .................................................. 40
Figure 11: Plot of absolute pressure vs position for MATLAB model and different ANSYS Fluent
models on the concave trapezoidal geometry at $M = 4.24$. .................................................. 41
Figure 12: Plot of absolute pressure vs position for MATLAB model (100 points) and different
ANSYS Fluent models on the convex arc geometry at $M = 4.24$. ......................................... 42
Figure 13: Plot of absolute pressure vs position for MATLAB model using various numbers of
points and ANSYS Fluent inviscid model on the convex arc geometry at $M = 4.24$. .......... 43
Figure 14: Plot of absolute pressure maximums and minimums vs Mach number for MATLAB
model (5 points) and different ANSYS Fluent models on the convex trapezoidal geometry. ..... 44
Figure 15: Plot of absolute pressure maximums and minimums vs Mach number for MATLAB
model (5 points) and different ANSYS Fluent models on the concave trapezoidal geometry. .... 45
Figure 16: Plot of absolute pressure maximums and minimums vs Mach number for MATLAB
model (5 points) and different ANSYS Fluent models on the convex arc geometry. ............... 45
Figure 17: Plot of absolute pressure maximums and minimums vs Mach number for MATLAB
model (5 points) and different ANSYS Fluent models on the concave arc geometry. ............... 46
Figure 18: Plot of absolute pressure vs horizontal position for the ANSYS Fluent laminar and $k - 
\varepsilon$ models using constant and temperature dependent properties on the convex trapezoidal geometry
at $M = 4.24$. ...................................................................................................................... 47
Figure 19: Plot of temperature vs height above the plate for the ANSYS Fluent laminar and $k - \epsilon$ models using constant and temperature dependent properties on the convex trapezoidal geometry at $M = 4.24$. ................................................................. 48

Figure 20: Plot of (A) the top panel surface temperature profile vs position and of (B) the top panel surface heat flux profile vs position from Thornton and Dechaumphai (1988). .................. 53

Figure 21: Plot of (A) the top panel surface temperature profile vs position and of (B) the top panel surface heat flux profile vs position for ANSYS Fluent simulations. ......................... 53

Figure 22: Plot of the top panel surface temperature profile vs position for ANSYS Fluent simulations at $t = 10$ s for different transfer intervals. .................................................. 57

Figure 23: Plot of the top panel surface heat flux profile vs position for ANSYS Fluent simulations at $t = 10$ s for different transfer intervals. .................................................. 58

Figure 24: Plots of the density fields at $t = 30$ s for (A) Thornton and Dechaumphai (1988) and (B) ANSYS Fluent simulations for convex (top) and concave (bottom) panel setups. .......... 62

Figure 25: Plot of the top panel surface temperature vs horizontal position for ANSYS Fluent laminar simulations for the pinned support panel and convex panel setup at $t = 10$ s following the deformation update by UDF. .......................................................... 64

Figure 26: MATLAB program procedure flow diagram. .......................................................... 72

Figure 27: Images of (A) mesh A and (B) mesh B.......................................................... 73

Figure 28: Plot of the top panel surface temperature profile vs position at time increments of $t = 10, 20, \text{ and } 30$ s for convex boundary conditions (SI units). .......................................................... 74

Figure 29: Plot of the top panel surface heat flux profile vs position at time increments of $t = 10, 20, \text{ and } 30$ s for convex boundary conditions (SI units). .......................................................... 74

Figure 30: Plot of static pressure at times of $t = 10$ s (A), 20 s (B), and 30 s (C) for the ANSYS Fluent laminar solver with convex boundary conditions. .................................................. 75

Figure 31: Plot of the top panel surface heat flux profile vs position for the ANSYS Fluent laminar, $k - \epsilon$, and $k - \omega$ models for convex boundary conditions at $t = 2.5$ s. .................. 76

Figure 32: Plot of the top panel surface temperature profile vs position for the ANSYS Fluent laminar, $k - \epsilon$, and $k - \omega$ models for convex boundary conditions at $t = 2.5$ s .................. 76

Figure 33: Plot of (A) the top panel surface temperature profile vs position and of (B) the top panel surface heat flux profile vs position for ANSYS Fluent laminar simulation at $t = 10$ s with the walls preceding and following the panel having constant heat flux $q = 43,720 \text{ W/m}^2$. ..... 77
Figure 34: Plot of (A) the top panel surface temperature profile vs position and of (B) the top panel surface heat flux profile vs position for ANSYS Fluent laminar simulation at $t = 10$ s with the inlet having a predefined temperature profile. ................................................................. 78

Figure 35: Panel thermal eccentric loading diagram. ................................................................. 80

Figure 36: Plot of thermal conductivity vs temperature for Prandtl number and kinetic theory formulations. .............................................................................................................. 87

Figure 37: Plot of (A) empirical Prandtl number data vs temperature and of (B) error between $Pr = 0.72$ assumption and empirical Prandtl number data vs temperature. ................................. 87

Figure 38: Plot of von Mises stress vs horizontal position for the top and bottom surface of panel for loosely coupled simulation at $t = 30$ s for the ANSYS Fluent laminar solver. ................. 89
Nomenclature

\( A \) Panel Cross Sectional Area \((m^2)\)
\( A_{ik} \) POD Eigenvalue solution \((None)\)
\( a_i \) POD Mode Amplitudes \((None)\)
\( \alpha \) Coefficient of Thermal Expansion \((K^{-1})\)
\( B \) Empirical Integration Constant \((None)\)
\( \beta \) Shock Angle \((rad)\)
\( C \) Buckling End Condition Constant \((None)\)
\( C(t, t_k) \) POD Tensor \((None)\)
\( c \) Local Speed of Sound \(\left(\frac{m}{s}\right)\)
\( c_p \) Specific Heat at Constant Pressure \(\left(\frac{J}{kg-K}\right)\)
\( c_v \) Specific Heat at Constant Volume \(\left(\frac{J}{kg-K}\right)\)
\( \chi \) Explicit Shock Angle Equation Secondary Placeholder Variable \((None)\)
\( d \) Panel Deflection \((m)\)
\( \delta \) Strong/Weak Shock Solution Parameter \((None)\)
\( e \) Internal Energy \(\left(\frac{J}{kg}\right)\)
\( e_{load} \) Loading Eccentricity \((m)\)
\( E \) Modulus of Elasticity \((Pa)\)
\( E_{flow} \) Flow Energy \(\left(\frac{J}{kg}\right)\)
\( \epsilon \) Emissivity \((None)\)
\( \gamma \) Specific Heat Ratio \((None)\)
\( h \) Enthalpy \(\left(\frac{J}{kg}\right)\)
\( I \) Second Moment of Area \((m^4)\)
\( k \) Thermal Conductivity \(\left(\frac{W}{m-K}\right)\)
\( k_{energy} \) Turbulent Kinetic Energy \(\left(\frac{J}{kg}\right)\)
\( k_{gyr} \) Radius of Gyration \((m)\)
\( k_{tur} \) Turbulent Thermal Conductivity \(\left(\frac{W}{m-K}\right)\)
\( \kappa \) Von Karman Constant \((None)\)
\( L \) Horizontal Length \((m)\)
\( L_{pan} \) Panel Length \((m)\)
\( \lambda \) Explicit Shock Angle Equation Primary Placeholder Variable \((None)\)
\( M \) Mach Number \((None)\)
\( M_w \) Molecular Weight \(\left(\frac{kg}{mol}\right)\)
\( M_\# \) Number of POD Snapshots \((None)\)
\( \mu \) Dynamic Viscosity \(\left(\frac{kg}{m-s}\right)\)
\( \mu_1 \) Forward Mach Angle \((rad)\)
\( \mu_2 \) Rearward Mach Angle \((rad)\)
\( N_\# \) Number of POD Modes \((None)\)
\( \nabla \) Gradient Operator \((None)\)

\( \nu \) Kinematic Viscosity \(\text{m}^2\text{s}^{-1}\)

\( u(M) \) Prandtl Meyer Function \((\text{rad})\)

\( Pr \) Prandtl Number \((None)\)

\( p \) Pressure \((\text{Pa})\)

\( p_{\text{crit}} \) Critical Pressure \((\text{Pa})\)

\( p_r \) Reduced Pressure \((None)\)

\( p_0 \) Static Pressure \((\text{Pa})\)

\( \varphi_i(\vec{x}) \) POD Modes \((None)\)

\( q \) Heat Flux \(\text{w/m}^2\)

\( R \) Universal Gas Constant \(\frac{J}{\text{mol-K}}\)

\( Re \) Reynolds Number \((None)\)

\( \rho \) Density \(\text{kg/m}^3\)

\( \rho_0 \) Static Density \(\text{kg/m}^3\)

\( \sigma \) Stefan-Boltzmann Constant \(\frac{W}{\text{m}^2\text{K}^4}\)

\( \sigma_{\text{yield}} \) Yield Stress \((\text{Pa})\)

\( t \) Time (s)

\( t_{\text{pan}} \) Panel Thickness (m)

\( T \) Temperature (K)

\( T_{\text{crit}} \) Critical Temperature (K)

\( T_r \) Reduced Temperature \((None)\)

\( T_0 \) Static Temperature (K)

\( \vec{\tau} \) Stress Tensor \((\text{Pa})\)

\( \tau_w \) Wall Stress \((\text{Pa})\)

\( \theta \) Turn Angle \((\text{rad})\)

\( u \) Horizontal Velocity \(\text{m/s}\)

\( \vec{u} \) Velocity Vector \(\text{m/s}\)

\( \vec{u}' \) Velocity Fluctuation Vector \(\text{m/s}\)

\( \vec{\mu}' \) Normalized Velocity Fluctuation Vector \((None)\)

\( \vec{\mu} \) Mean Velocity Vector \(\text{m/s}\)

\( u_* \) Friction Velocity \(\text{m/s}\)

\( u_+ \) Normalized Boundary Layer Velocity \((None)\)

\( v \) Vertical Velocity \(\text{m/s}\)

\( w_{\text{pan}} \) Panel Width (m)

\( x \) Horizontal Position (m)

\( y \) Vertical Position (m)

\( y_+ \) Normalized Boundary Layer Vertical Distance \((None)\)
Dedication

This thesis is dedicated to all of the professors, classmates, friends, and family who have supported me over the past few years. Thank you for everything!
Acknowledgements

This work was completed under the guidance of Dr. Edgar Caraballo. Great appreciation is felt towards Dr. Amit Shukla and Dr. Andrew Sommers for being on the thesis committee and contributing valuable input. Likewise, thank you to Dr. James VanKuren and Dr. James Moller for all of your advice. Special thanks to the Ohio Space Grant Consortium (OSGC) from NASA for their financial support through the OSCG grant. Thank you to all my professors and fellow classmates who helped me learn and enjoy my time at Miami University. I would like to thank my parents and family for their continued support throughout the duration of my education. Lastly, I would like to thank Jesus Christ my God for His love in my life.
Chapter 1: Literature Review

1.1 Introduction

Hypersonic flight is a flow regime at speeds near and greater than five times the speed of sound, where several phenomena become important. According to Anderson (2003) these include severe heating from friction and chemical breakdown of the molecules in the air. Experimental data is often hard to attain as hypersonic wind tunnels and aircraft are expensive to build and operate. Computational fluid dynamics (CFD) is often used as an alternative, and allows data to be generated rather inexpensively by iteratively solving the Navier-Stokes equations that describe the fundamental behavior of fluid flow.

Computational methods can be particularly helpful in analyzing and predicting when airfoil panel buckling will occur and how to prevent it. When the airfoil of a hypersonic aircraft is dramatically heated, the panel surface can expand against itself causing internal stresses and eventual buckling. Buckling manifests itself as the surface bulging outward or dipping inward. These deflections can occur rapidly during flight leading to oscillatory behavior, or simply result in a single, sudden deflection event followed by smaller subsequent changes. Either way, these deflections affect the airflow near the surface by changing the shock wave and expansion wave behavior along with air properties such as temperature, pressure, and density. This complex interplay between the air and the panel is difficult to model with CFD as the fluid and structure domains are physically coupled.

Previous research has approached this coupled system in different ways. The simplest method is to uncouple the system and assume predetermined thermal conditions at the air-panel interface while focusing on the structural behavior. Conversely, simulations can be fully coupled where the fluid and structural solver exchange information every time step for greater accuracy, but at increased computational cost. Loosely coupled simulations offer a compromise between accuracy and efficiency by only transferring data at certain time intervals. Loosely coupled simulation was the method chosen for this study.

However, there are still computational costs associated with loosely coupled simulations, especially for larger models. Reduced order modeling (ROM) can be constructed from existing data sets to simulate a scenario, sacrificing some accuracy, but vastly reducing computational costs.
This study examined an earlier study as a starting point in order to develop loosely coupled simulations using CFD and finite element structural analysis. The effect of transfer interval sizes was examined along with different theoretical CFD models. Data saved from these simulations is available for creation of a ROM via proper orthogonal decomposition (POD).

1.2 Background

All CFD software seeks to solve the Navier-Stokes equations numerically, as they are complicated partial differential equations that generally have no analytical solutions. Various numerical techniques exist including the finite volume, finite element, and finite difference methods. Work by Jeong and Seong (2014) recommends the finite volume method for CFD simulations to improve accuracy, improve efficiency, and reduce solution dependence upon mesh shape.

Many research groups have done CFD work for hypersonic flows using various approaches. A review by Sinha (2010) highlights some of the phenomena that should be considered in the hypersonic region such as chemical reactions in the air and strong gradients in the flow field variables. However, according to Anderson (2003), temperatures are low enough for this study to safely neglect chemical reactions in the air as air typically does not disassociate until it reaches approximately 2,500 K. Only ideal gas and real gas effects were considered.

Effects in the boundary layer also needed to be considered. A study by Stemmer and Adams (2009) takes a detailed look at the boundary layer of hypersonic flow over a ramp using the large eddy simulation (LES) turbulence model. Their simulations compared the results of flows using ideal gas methods of density and those including real gas effects. According to the study, complex shock wave/boundary layer interaction requires the use of a very fine grid near walls in order to capture phenomena accurately. Care must be taken to ensure that enough grid layers are placed within the actual boundary layer thickness or artificial phenomena can be introduced. These grid refinements add to the computational cost, but are necessary for hypersonic flow and were used in this study.

The fluid field is not the only area of interest in this study. The physical response of panels in an airfoil was also desired. This response generally manifests itself as panel thermal buckling or flutter. Airfoil panel buckling due to aerodynamic heating has been examined for several decades. Before the advent of numerical CFD, a study by Mansfield (1960) sought to analytically
determine buckling forces for simplified models of airfoil panels. More recent studies have largely moved on to numerical analyses of buckling.

Two NASA research groups led by Ko (1995; 2004) directly examined buckling behavior of sandwich panels using finite element methods and a minimum-potential energy buckling method. The first study sought to determine buckling temperatures for different fixed and pinned boundary conditions. These yielded several mode shapes and the approximate temperatures at which they occur. While the sandwich panels are a bit more complex than the solid panel in this study, the qualitative mode shapes are useful references for general trends. The second study examined buckling behavior of panels based upon pre-determined temperature distributions. This approach improves upon buckling accuracy as the temperature distribution in panels in an airfoil is unlikely to be constant. These studies offer potential to provide comparison for trends in buckling results. They also support finite element methods as appropriate techniques for thermal buckling analysis.

In reality, the flow and panel behaviors are not independent and interact in a complex manner. This is compounded by the fact that the thermal boundary conditions for the panels range widely and are often not accurately known. Coupled fluid-thermal-structural simulations therefore are needed to accurately capture all the phenomena present near a panel in hypersonic flow. A study by Oppattaiyamath, Reddy, Jammy and Kulkarni (2013) ran thermally coupled simulations to examine the differences between various panel boundary conditions in a hypersonic flow. Although panel deformation resulting from the temperature change was not accounted for, the temperature distributions are a useful reference.

Studies on fully coupled fluid-thermal-structural interactions have also been done. A study by Ma et al. (2012) discusses the various coupling systems found in typical hypersonic flow including aero-thermo, aero-elastic, thermo-elastic, aero-control, and structure-control coupling. The terms aero, thermal, elastic, control, and structure stand for the gas flow, heat transfer, structure deformation, flight control, and controlled structure deformation respectively. Each of these systems has its own set of coupling equations. All of these equations must be solved simultaneously or iteratively in order to correctly predict hypersonic flow fields. Ma et al. (2012) developed an eight step simulation method in order to solve the equations. While the exact process is not explained, designing such a computational method would be quite difficult. Another study by Wan et al. (2014) introduced a new coupling scheme based off hypersonic lifting surface theory,
hypersonic heating theory, and traditional structural analysis. This technique forgoes traditional CFD for a more analytic approach. The team analyzed surface temperatures and flutter, and found the method to be quite accurate. However, complexity remains an issue.

This complexity can be reduced through loosely coupled simulations, but accuracy must be assessed. The technique first solves the fluid field and convective heat transfer, transfers the thermal results to a structural solver, and then finds the resulting structural response. The subsequent deformations are finally input back into the fluid domain. This process is repeated iteratively throughout the duration of the simulation. A study by Culler and McNamara (2011) examined the differences between fully and partially coupling the flow field and structure on a typical aircraft panel in hypersonic flow. They concluded that the two methods can yield fairly different results, especially in regards to predicting buckling behavior and location of temperature maximums along the panel. Such behavior was part of the motivation in analyzing the effect of transfer interval sizes for the loosely coupled simulations.

1.3 Case Study: Thornton and Dechaumphai (1988)

One study led by Thornton and Dechaumphai (1988) is especially relevant to this study as it offers baseline methods and results that serve as a starting point for this work. The paper includes simulation data for a flat panel fixed at either end. Both concave and convex deformation were considered along with the panel boundary conditions necessary for their initiation. Flow was assumed to be two-dimensional. The study examined Mach 6.57 flow over a sharp-edged panel holder, which caused an oblique shock and a drop to Mach 4.24, fully developed flow which was seen by the panel. The considered domain was slightly larger than the panel and had a fully developed, Mach 4.24 inlet velocity condition with the Reynolds number per length being \( \frac{Re}{L} = 2.17 \times 10^6 \ m^{-1} \). Neither the initial Reynolds number nor the flow length preceding the domain were mentioned.

A loosely coupled fluid-thermal-structural approach was taken for the study. The fluid simulation was run for \( 4 \times 10^{-4} \ s \) through 4,000 time steps of \( 1 \times 10^{-7} \ s \) each at which point the fluid reached essentially steady state conditions. A thermal simulation was then run for \( 10 \ s \) using 1,000 times steps of \( 1 \times 10^{-2} \ s \) each to determine the temperature distribution of the panel. This temperature distribution was then fed into a quasi-static, finite element structural simulation to determine the panel deflection. The process was repeated three times until results for a full \( 30 \ s \)
were attained. A viscous, laminar, finite element fluid solver was used. The validity of this solver will be discussed.

The simulations in this study were based off the domain and methodology used by Thornton and Dechaumphai (1988). The two-dimensional approach was kept by assuming the third dimension to be semi-infinite as would be approximated by a long wing. A study by Deepak, Gai, and Neely (2013) used the same assumption for compression corners at hypervelocity speeds (approximately Mach 8 and higher) and reported that experimental flow was nominally two-dimensional at the mid-span of the geometry.

1.4 Hypersonic Fluid Dynamics

CFD operates by solving a system of equations including the Navier-Stokes equations, nonlinear partial differential equations that have no general analytical solutions. The first equation is the continuity equation given as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0$$  \hspace{1cm} (1)

where $\rho$ is the density, $\vec{u}$ is the velocity vector, $t$ is the time, and $\nabla$ is the gradient operator.

The second portion are the Navier-Stokes equations, which consider the conservation of linear momentum given as

$$\frac{\partial}{\partial t} (\rho \vec{u}) + \vec{u} \cdot \nabla (\rho \vec{u}) = -\nabla p + \nabla \cdot (\vec{\tau})$$  \hspace{1cm} (2)

where $p$ is the pressure, and $\vec{\tau}$ is the stress tensor. As this study is two-dimensional, only the horizontal and vertical linear momentum equations will be considered.

The final equation solved along with the Navier-Stokes equations is the energy conservation equation given as

$$\frac{\partial}{\partial t} (\rho E_{flow}) + \nabla \cdot (\vec{u} (\rho E_{flow} + p)) = \nabla \cdot (k_{eff} \nabla T + (\tau_{eff} \cdot \vec{u}))$$  \hspace{1cm} (3)
where $E_{\text{flow}}$ is the flow energy and $k_{\text{eff}}$ is the effective thermal conductivity of the flow. The term $\bar{\tau}_{\text{eff}} \cdot \bar{u}$ accounts for viscous dissipation making the energy equation necessary for hypersonic, compressible flow.

Fluid flow is often characterized by the Reynolds number which is defined by White (2011) as the ratio of the inertial to viscous forces. For a flat plate, it is convenient to write the Reynolds number per length, which for this study at initial conditions of 295 $K$ is

$$\frac{Re}{L} = \frac{\rho u}{\mu} = \frac{(1.198 \text{ kg/m}^3)(1443 \text{ m/s})}{(1.821 \times 10^{-5} \text{ kg/m/s})} = 9.493 \times 10^7 \text{ m}^{-1}$$

The Reynolds number typically is used to label a flow as laminar or turbulent. Durbin and Medic (2007) define turbulent flow as containing fluctuations deviating from the mean velocity. These fluctuations are defined below as

$$\bar{\bar{u}}' = \bar{u} - \bar{\bar{u}}$$

where $\bar{\bar{u}}$ is the instantaneous velocity and $\bar{\bar{u}}$ is the mean velocity. In comparison, laminar flow is the regime where there are minimal velocity fluctuations ($\bar{\bar{u}}' = 0$) and therefore the instantaneous velocity is always equal to the mean velocity ($\bar{u} = \bar{\bar{u}}$). The Navier-Stokes equations can be solved using either the instantaneous or mean velocity and will yield the same result. This is easy to understand with laminar flow as the two velocities are equivalent. Turbulent flow is a bit different. Directly solving using the instantaneous velocity is called direct numerical simulation (DNS) and requires using appropriately small time scales and length scales. However, this is often computationally prohibitive. An alternative approach uses the mean velocity and results in the Reynolds-averaged velocity equations (RANS) where the velocity fluctuations become important, and yield an extra term, the Reynolds stress term given below as

$$-\rho \bar{u}' \bar{\bar{u}}'$$
This Reynolds stress is found within the stress tensor \( \bar{\tau} \) and introduces extra unknowns without providing extra equations with which to solve them. Closure to these RANS equations generally uses statistical averaging over many samples. Various turbulence models handle this averaging in different ways, but tend to combine a mixture of theory and empirical observations. Often, models will incorporate the turbulent kinetic energy per unit mass defined as

\[
k_{\text{energy}} = \frac{1}{2}(u'\bar{u} + v'\bar{v})
\]  

(7)

where \( u' \) and \( v' \) are the two spatial components of the average velocity fluctuation. Turbulent kinetic energy is used to predict the amplitude of the velocity fluctuations that allow turbulence to be modeled. Accurate turbulence modeling becomes increasingly important at higher Mach numbers as turbulence can be responsible for a large portion of the heat transfer resulting from viscous heating.

As previously stated, Thornton and Dechaumphai (1988) used a viscous, laminar solver in their study. If the flow field were fully laminar, a laminar solver would certainly be justified. Unfortunately, it is unclear in their paper what the Reynolds number was at the inlet of the domain or what the flow length prior to the domain was. For a flat plate, White states that turbulence typically initiates for a critical Reynolds number on the order of \( Re_{\text{crit}} = 5 \times 10^5 \). At an air temperature of 295 K and a flow velocity of 1443 m/s, air has a Reynolds number per length of \( \frac{Re}{L} = 9.493 \times 10^7 \text{ m}^{-1} \) from Equation (4). Combining this with the aforementioned critical Reynolds number yields the initiation of turbulence at a length

\[
L_{\text{crit}} = \frac{Re_{\text{crit}}}{\frac{Re}{L}} = \frac{5 \times 10^5}{9.493 \times 10^7 \text{ m}^{-1}} = 5.267 \times 10^{-3} \text{ m}
\]

(8)

The domain begins with fully developed flow at the inlet based on conditions given from \( M = 6.57 \) flow striking an angled, sharp leading edge, producing an oblique shock to drop to \( M = 4.24 \) flow, and then flowing over a wall for some unknown distance. Although not drawn to scale, a schematic shown by Thornton and Dechaumphai (1988) seems to indicate that the pre-inlet length is at least as long as the panel. This would mean that the flow had moved over the surface
for at least 0.1016 m which would make the flow fully turbulent by the time it reached the inlet of the domain. Although partially speculative based on the lack of information, this computation indicates that the use of a laminar solver for this scenario might not be not entirely justified.

However, Anderson (1989) states that a value of \( Re_{crit} = 5 \times 10^5 \) is not a valid assumption for hypersonic flow in general. Practically, the critical Reynolds number of hypersonic flow is dependent upon many flow parameters, but could ultimately be on the order of \( Re_{crit} = 10^8 \). Such a value would yield a transition length of \( L_{crit} = 1.053 \) m, well beyond the length of the panel. This indicates that the flow would in fact still be laminar and a laminar solver would be the appropriate choice. As the exact critical Reynolds number could not be calculated, it cannot be definitely determined whether or not the flow in this scenario is turbulent or laminar, and simulations were run using the laminar model and the \( k - \epsilon \) turbulence model to provide comparison between the two.

Effects in the boundary layer also need to be given special consideration in the hypersonic regime. A computationally inexpensive approach traditionally employed to solve turbulent boundary layers involves wall functions which allow the boundary layer in the immediate vicinity to be resolved semi-empirically with a coarse mesh rather than with fine grid resolution. The derivation of one such wall function, the log-law as explained by Durbin and Medic (2007), is given below. A parameter called friction velocity, which approximately measures the order of magnitude of the fluctuations in turbulent velocity, is defined below as

\[
  u_* = \sqrt{\frac{\tau_w}{\rho}}
\]

with \( \rho \) representing the density and \( \tau_w \) being the viscous wall stress defined as

\[
  \tau_w = \mu \left. \frac{\partial u}{\partial y} \right|_0
\]

where \( \mu \) is the viscosity. Having friction velocity allows “\( + \)” units, dimensionless parameters describing distance from the wall and fluid velocity, to be defined respectively as
where $\nu$ is the kinematic viscosity. For zero pressure gradient boundary layers often found in hypersonic flight due to atmospheric conditions in front of and behind an aircraft, these parameters are found to be related by the log-law equation

$$u_+ = \frac{u}{u_*} \log(y_+) + B u_*$$

(13)

with $\kappa$ being the empirically found Von Karman constant and $B$ being an empirically found integration constant. If $u_*$ is known, the log-law can find $u_+$ for a given $y_+$ (and therefore for a given $y$), and determine the velocity at that point. This allows, and in fact requires, the first grid layer to be outside of the viscous sublayer layer, which reduces the overall number of grid points. This criterion is approximately met when $y_+ \geq 10$ for the $k - \epsilon$ model. This derivation is given generically for all turbulent flows, so its validity for hypersonic flow would have to be examined. Laminar flows have a simpler boundary layer profile and therefore do not use this method.

### 1.4.1. ANSYS Fluent Theory

ANSYS Fluent version 15.0 is a CFD program that uses the finite volume method to solve simulations. Various controls relating to fluid properties, solver type, turbulence models, and other fluid mechanics properties are available as selectable options depending upon the simulation being run. In order to correctly select these options, various sources were consulted along with the ANSYS Fluent user guide (2013). A brief summary of the theory behind ANSYS Fluent and the controls chosen for the simulations follows.

Like all CFD solvers, ANSYS Fluent uses various numerical methods to solve the Navier-Stokes equations. The finite volume method is the discretization method used in ANSYS Fluent, as it is the preferred formulation for CFD simulations according to Jeong and Seong (2014). A pressure-based and density-based solver are available. The coupled form of the pressure-based
solver first simultaneously solves the momentum equations and a pressure correction equation derived from the continuity equation. Mass fluxes are then updated using the pressure correction values. Finally, the energy equation and other scalars are found using the results of the previous computations. Conversely, the density-based solver simultaneously solves the continuity, momentum, and energy equations for their unknowns based on the solution from the previous step. Any additional unknown scalars including turbulence are then solved for using the newly calculated values. If a check shows that convergence has not occurred, the process repeats. As the flow regime of relevance is hypersonic, the density-based solver was chosen for this study according to ANSYS Fluent user guide (2013) recommendations. While it typically takes more memory than the pressure-based solver, the density-based solver tends to be more accurate for high-speed, compressible flows. ANSYS Fluent also offers an implicit and explicit density-based solver with the implicit solver being chosen to aide with convergence and stability.

Equations (1), (2), and (3) given above are similar to the form of the Navier-Stokes equations used by ANSYS Fluent. However, the term $k_{eff}$ in Equation (3) must receive further attention. $k_{eff}$ is the sum of the fluid’s thermal conductivity and the turbulent thermal conductivity $k_{tur}$, which depends on the chosen turbulence model. ANSYS Fluent offers several turbulence models for various situations including the Spalart-Allmaras, $k - \epsilon$, $k - \omega$, Transition $k - kl - \omega$, Transition SST, Reynolds Stress, and Scale-Adaptive Simulation (SAS) models along with the inviscid and laminar solvers which are non-turbulent. Turbulence modeling constitutes a large area of hypersonic flow research with many studies having been done on the topic comparing the different models. A review by Roy and Blottner (2006) examined several studies done on baseline hypersonic geometries and offers general recommendations for simulations. Of the turbulence models listed above, the paper noted that the Spalart-Allmaras, $k - \epsilon$, and $k - \omega$ models have been extensively studied and verified. As these models require that the flow be fully turbulent, the study highlights the importance of ensuring that turbulence is initiated prior to the application of the models and the difficulty of comparing results for turbulent flows if the upstream turbulent boundary layer thickness has not been reported. The study also stated that skin friction and surface heat flux predictions are not generally accurate for most commercial turbulence models. This fact is considered when examining buckling results.

Each turbulence model must be able to properly render the boundary layer and the grid near the wall must be optimized to facilitate this boundary layer computation. As stated above,
boundary layer properties can be found using the near-wall method or using wall functions. Traditionally, care must be taken to ensure that the grid spacing near the wall closely matches the criteria given for each method, with the near-wall method needing very fine resolution and wall functions needing the first grid layer to be sufficiently far away. Fortunately, turbulence models in ANSYS Fluent employ both boundary layer methods and are able to provide accurate results for any grid spacing. Since the grid spacing at the wall is not restricted by ANSYS Fluent, the near wall method could be used without depending upon using specific $y_+$ values. Instead, meshes were simply refined by placing as many grid layers as possible near walls to ensure that shock wave/boundary layer interactions are accurately resolved. This approach was especially relevant when using the laminar solver because wall functions are not applicable.

Even without considering the extra equations needed when including velocity fluctuations, the system of equations including the Navier-Stokes equations has one more unknown variable than equations, and one extra equation must be used to allow a solution. For this study, an ideal gas formulation is used to supply the last equation:

$$\rho = \frac{p}{R \cdot M_w \cdot T}$$  \hspace{1cm} (14)

where $p$ is the absolute pressure, $R$ is the universal gas constant, $M_w$ is the molecular weight, and $T$ is the temperature. The ANSYS Fluent user guide recommends the ideal gas density formulation for compressible flow. However, as hypersonic flow often deals with high-temperature and/or reacting gases, an ideal gas assumption must be justified by ensuring the air acts as a continuum and stays within a certain pressure and temperature range. According to the ANSYS Fluent user guide (2013) and Cengel and Boles’ thermodynamics textbook (2002), an ideal gas approximation holds if the reduced temperature is greater than two and the reduced pressure is less than one. The textbook further states that the critical temperature of air is $T_{crit} = 132.5 \, K$ and the critical pressure of air is $p_{crit} = 3.77 \times 10^6 \, Pa$. In addition, with an initial temperature of $T = 295 \, K$ and an initial atmospheric pressure of $P = 101,325 \, Pa$, basic steady state hypersonic simulations yield a minimum temperature of $T_{min} = 295 \, K$ and a maximum pressure of approximately $P_{max} = 1.60 \times 10^5 \, Pa$. The reduced temperature and pressure are then computed below as
\[ T_r = \frac{T_{\text{min}}}{T_{\text{crit}}} = \frac{295 \, K}{132.5 \, K} = 2.226 > 2 \]  
Equation (15)

\[ p_r = \frac{p_{\text{max}}}{p_{\text{crit}}} = \frac{1.60 \times 10^5 \, Pa}{3.77 \times 10^6 \, Pa} = 0.042 \ll 1 \]  
Equation (16)

As hypersonic flow typically produces temperatures much higher than the initial temperature, the reduced temperature should always be higher than two as long as temperatures remain greater than \( T = 265 \, K \). Similarly, the reduced pressure is so far below one that the maximum pressure should never be high enough in any scenario to be an issue. Therefore, it was concluded that using ideal gas law density is a valid approach for the hypersonic simulations in this study as long as the air molecules do not disassociate. However, Anderson (2003) reports that at atmospheric pressure, air does not disassociate below approximately 2,500 \( K \). To ensure this criterion was fully met, simulations were halted once temperatures reached about 1,000 \( K \).

Viscosity is a crucial parameter in hypersonic flow for determining heat generation. ANSYS Fluent offers several Newtonian fluid, temperature dependent models for calculating viscosity including the Sutherland formulation, the power-law formulation, and the kinetic theory formulation. This study will emulate the study by Thornton and Dechaumphai (1988) and use the Sutherland formulation with constant coefficients justified using Stokes’ hypothesis. Bush and Krishnamurthy (1998) state that the Sutherland formulation is more accurate over a wider range of temperatures in hypersonic flow than are power laws. Anderson (1989) reports that the Sutherland formulation is accurate over a temperature range of a few thousand degrees. Studies by Emanuel (1992) and Buresti (2015) support the use of Stokes’ hypothesis without much loss of accuracy for gases such as air with a low viscosity ratio. Blackaby, Cowley, and Hall (1993) and Deepak, Gai, and Neely (2013) further support that the Sutherland formulation is accepted for use in high temperature hypersonic flows.

Thermal conductivity is another important parameter for hypersonic flows as it helps determine heat flow through the air. Thornton and Dechaumphai (1988) reported using a temperature dependent formulation based on a constant Prandtl number \( Pr = 0.72 \) for thermal conductivity, but do not specify the exact formulation. It is assumed they used the definition of
the Prandtl number to calculate the thermal conductivity, a formulation also used by Deepak, Gai, and Neely (2013) and Anderson (1989) given below as

\[ k = \frac{c_p \mu}{Pr} \]  

(17)

where \( c_p \) is the specific heat at a constant pressure.

The only temperature dependent thermal conductivity options offered by ANSYS Fluent are those requiring empirical inputs and kinetic theory. As no empirical data was available for this study Equation (17) was chosen. Kinetic theory was also considered as Figure 36 in Appendix F shows that the kinetic theory formulation and the Prandtl number definition give similar results up to 1,000 \( K \). However, the validity of kinetic theory was ultimately unable to be verified leading to the Prandtl number formulation being implemented into a user defined function (UDF) in ANSYS Fluent. This choice also seemed the most likely to replicate the formulation used by Thornton and Dechaumphai (1988). In addition, when compared to empirical Prandtl number data from a heat transfer textbook by Incropera et al. (2003), a constant value of \( Pr = 0.72 \) is accurate to within 6 % from \( T = 250 K \) to \( T = 1,000 K \) giving a close agreement to data. This data comparison can be seen in Figure 37 in Appendix F.

Thornton and Dechaumphai (1988) likewise did not specify the temperature dependent formulation used for specific heat. ANSYS Fluent also only includes empirical options and kinetic theory for calculating the specific heat of air at different temperatures. However, Cengel and Boles’ thermodynamics textbook (2002) lists calculated values of the specific heat for air that are only temperature dependent, allowing a linear interpolation table to be used in ANSYS Fluent. These values extend from 250 \( K \) to 1,000 \( K \) and can be seen in Table 17 in Appendix H.

Sink radiation, or radiation to an infinite medium, is also considered in this study to account for radiation to space. Sink radiation can be specified as a thermal boundary condition in ANSYS Fluent and the resulting radiative heat flux is given below as

\[ q = \varepsilon \sigma (T^4 - T_\infty^4) \]  

(18)
where $\epsilon$ is the external emissivity, $\sigma$ is the Stefan-Boltzmann constant, $T_\infty$ is the external temperature, and $T$ is the temperature of the surface to which the boundary condition is applied. For this study, $\sigma = 5.67 \times 10^{-8} \frac{W}{m^2-K^4}$, $\epsilon = 1$ approximating space as a perfect black body, and $T_\infty = 1 \, K$ to approximate the low temperatures of space.

### 1.4.2. Supersonic Oblique Shock Equations

Although the formulations and assumptions used in ANSYS Fluent have been well documented above, the results were checked against theory to ensure realistic simulations. Approximate solutions for simple supersonic and hypersonic flow fields can often be found using theoretical oblique shock equations. These equations can be solved explicitly and without CFD software, but are only valid for simple geometries and certain conditions. For this study, the oblique shock equations were used to verify steady state solutions found using ANSYS Fluent. Anderson’s book (2003) on compressible flow was used as a reference for the oblique shock equations.

Supersonic flow can include normal shock waves, oblique shock waves, and expansion waves. Normal shock waves are perpendicular to the flow velocity and result in subsonic conditions behind the shock. Oblique shock waves occur at geometric compression corners of angle $\theta$ and appear at the shock angle $\beta$. Conditions are typically still supersonic behind oblique shocks. Expansion waves are the last type, and occur at geometric expansion corners of angle $\theta$. Expansion waves consist of a fan of infinite Mach waves bounded between forward and rearward Mach lines at angles $\mu_1$ and $\mu_2$ respectively. Each Mach wave is completely isentropic. A schematic of the three types of waves in supersonic flow can be seen below in Figure 1.

**Figure 1**: Diagram of a normal shock, oblique shock, and expansion wave.
Several assumptions are made in the derivation of the oblique shock equations and must be taken into account when comparisons are made. The equations assume that a calorically perfect gas is being modeled using the ideal gas law with constant specific heats $c_p$ and $c_v$. According to Anderson (2003), constant specific heat is a reasonably accurate assumption for non-chemically reacting flows under 1,000 $K$ with Mach numbers less than about five. Accuracy does begin to diminish in the hypersonic regime, but as the temperatures from the simulations in this study are below 1,000 $K$, these inaccuracies are judged to be mild. The constant specific heat requirement is adjusted later for loosely coupled simulations. The internal energy $e$ and enthalpy $h$ for a calorically perfect gas are calculated below in Equations (19) and (20)

$$e = c_v T$$  \hspace{1cm} (19)  \\
$$h = c_p T$$  \hspace{1cm} (20)

where $c_v$ is the specific heat at a constant volume.

The flow is also assumed to be isentropic and adiabatic everywhere except across shock waves as they are irreversible. Expansion waves around a sharp corner are isentropic by nature, and so fit this idealization well. Adiabatic flow, or flow without heat addition, corresponds to the isentropic assumption and is a reasonable approximation for the flow field away from surfaces. However, viscosity within the boundary layer leads to large amounts of heat addition in hypersonic flows near surfaces. Oblique shock theory ignores this effect leading to an adiabatic, inviscid flow assumption.

The oblique shock equations can describe steady state flows over compression and expansion corners. The corner angles for the given geometry are used to calculate the shock wave or expansion wave angles along with the change in flow variables including Mach number, pressure, temperature, and density. Equation (21) describes the shock angle $\beta$ for supersonic flow through a compression corner of angle $\theta$ as

$$\tan(\theta) = 2\cot(\beta) \frac{(M_1^2 \sin^2 \beta - 1)}{M_1^2(\gamma + \cos 2\beta) + 2}$$  \hspace{1cm} (21)
where $\gamma$ is the specific heat ratio and $M_1$ is the Mach number of the flow preceding the compression corner. As this equation is implicit and requires iterative techniques to solve, an explicit formulation of this equation found in Anderson was used for direct computation of $\beta$

$$
\tan(\beta) = \frac{M_1^2 - 1 + 2\lambda\cos[(4\pi\delta + \cos^{-1}\chi)/3]}{3\left(1 + \frac{\gamma - 1}{2}M_1^2\right)\tan(\theta)}
$$

where $\delta = 0$ is used for the strong shock solution and $\delta = 1$ for the weak shock solution. $\lambda$ and $\chi$ are placeholder variables given below as

$$
\lambda = \sqrt{(M_1^2 - 1)^2 - 3\left(1 + \frac{\gamma - 1}{2}M_1^2\right)\left(1 + \frac{\gamma + 1}{2}M_1^2\right)\tan^2(\theta)}
$$

$$
\chi = \frac{(M_1^2 - 1)^3 - 9\left(1 + \frac{\gamma - 1}{2}M_1^2\right)\left(1 + \frac{\gamma - 1}{2}M_1^2 + \frac{\gamma + 1}{4}M_1^4\right)\tan^2(\theta)}{\lambda^3}
$$

The weak shock solution is assumed here for analysis as external hypersonic flows rarely exhibit strong shock behavior. Anderson (2003) notes that the strong shock solution only manifests itself in the presence of strong, independent backpressure downstream of the shock. As the geometries in this study are mostly parallel to the flow, the upstream and downstream pressure are both approximately equivalent to atmospheric pressure, leading to the weak shock solution.

Care must be exercised when using Equation (22). Under certain conditions, $\beta$ will be an imaginary number. Such situations occur when the turn angle $\theta$ becomes too large for the Mach number and the oblique shock detaches from the compression angle forming a detached bow shock. Detached shocks are out of the scope of oblique shock theory and must be handled with CFD. Thus, any flow fields causing detached shock waves were not analyzed with oblique shock theory.

Expansion waves must be handled differently from shock waves as the airflow is being expanded rather than compressed. Additionally, expansion waves cause no change in entropy and
are therefore isentropic. The Prandtl-Meyer relation is used to find the angle of an expansion wave and the change in Mach number around an expansion corner by

\[ u(M) = \frac{y+1}{y-1} \tan^{-1} \left( \frac{y-1}{y+1} (M^2 - 1) \right) - \tan^{-1} \sqrt{(M^2 - 1)} \]  

(25)

where \( u(M) \) is called the Prandtl-Meyer function. Inserting \( M_1 \), the Mach number before the expansion corner, into the relation yields \( u(M_1) \), the Prandtl-Meyer function before the corner.

Once \( u(M_1) \) is found, one can solve for \( u(M_2) \), the Prandtl-Meyer function after the corner, using

\[ u(M_2) = \theta + u(M_1) \]  

(26)

Having been found, \( u(M_2) \) can be used to iteratively solve for \( M_2 \), the Mach number after the expansion corner from Equation (25). Once \( M_1 \) and \( M_2 \) are known, the angles of the forward and rearward Mach line can be found by

\[ \mu_{1,2} = \sin^{-1} \frac{1}{M_{1,2}} \]  

(27)

The angles \( \mu_1 \) and \( \mu_2 \) indicate the angular limits of the expansion wave through which the flow isentropically expands in an infinite number of Mach waves.

There also exist situations where the Prandtl-Meyer function fails. Flow at a given Mach number has a maximum turn angle \( \theta \) through which it can expand. Calculating \( v(M_2 = \infty) \) yields a maximum value for the Prandtl-meyer function. If the turn angle \( \theta > v(M_2 = \infty) \), Equation (26) will yield \( v(M_1) < 0 \) which is unphysical. In this scenario, the flow does not expand smoothly around the expansion corner and requires complex analysis. Mach numbers and geometries were chosen to avoid this situation.

The oblique shock equations also include formulas that define ratios across shock waves and expansion waves. The ratios across expansion waves for pressure, density, and temperature can be seen below respectively as
According to Anderson (2003), because expansion waves are completely isentropic, the static temperature, static pressure, and static density ratios across an expansion wave always equal unity because those three variables do not change under isentropic conditions. This is realistic for sharp corners, but is less applicable for expansion waves around curved corners. This assumption will be considered when analyzing curved geometry.

The definitions for these ratios across shock waves are different as isentropic conditions do not apply. The equations for the pressure, density, and temperature ratios across shock waves are given as

\[
\frac{p_2}{p_1} = 1 + \frac{2\gamma}{\gamma + 1} \left( M_1^2 \sin^2 \beta - 1 \right)
\]

\[
\frac{\rho_2}{\rho_1} = \frac{(\gamma + 1) M_1^2 \sin^2 \beta}{(\gamma - 1) M_1^2 \sin^2 \beta + 2}
\]

\[
\frac{T_2}{T_1} = \frac{p_2 \rho_1}{p_1 \rho_2} = \left[ 1 + \frac{2\gamma}{\gamma + 1} (M_1^2 \sin^2 \beta - 1) \right] \left[ \frac{(\gamma - 1) M_1^2 \sin^2 \beta + 2}{(\gamma + 1) M_1^2 \sin^2 \beta} \right]
\]

The Mach number after an oblique shock wave can also be described as
\[ M_2 = \sin(\beta - \theta) \sqrt{\frac{M_1^2 \sin^2 \beta + \left(\frac{2}{\gamma - 1}\right)}{\frac{2\gamma}{(\gamma - 1)}M_1^2 \sin^2 \beta - 1}} \]  

The last set of relationships that make up the oblique shock equations relate the static pressure, static density, and static temperature to the pressure, density, and temperature respectively at any given point in the flow field

\[
\frac{p_0}{p} = 1 + \frac{\gamma - 1}{2} M^2 
\]  

(35)

\[
\frac{\rho_0}{\rho} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\frac{1}{\gamma - 1}} 
\]  

(36)

\[
\frac{T_0}{T} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\frac{\gamma}{\gamma - 1}} 
\]  

(37)

All of these oblique shock equations allow the flow field to be determined for a simple geometry and were implemented in a simple MATLAB model.

1.5 Panel Deformation and Buckling

1.5.1 Elastic Deformation

Structural calculations for the panel also had to be considered. For the boundary conditions used in the study by Thornton and Dechaumphai (1988), no buckling was considered. Therefore, this study was able to use standard FEA for most of the structural analysis used in the loosely coupled simulations. Two-dimensional simulations were used with plane strain conditions where strain along the third axis was taken to be non-existent due to the assumption that the third dimension was infinite. While Thornton and Dechaumphai (1988) used large strain deformation with plasticity considered, this study only considered elastic deformation as no stress-strain plasticity data were available. The validity of this assumption was explored and confirmed by
comparing the maximum von Mises stress in the panel with the yield stress of the material. Further discussion can be found in section 3.2.1.

1.5.2. Buckling

Although panel buckling can often be a complex three-dimensional problem, the two-dimensional assumptions made for this study allows the panel to be approximated as a simple column. Column buckling can be predicted using Euler buckling theory which is detailed below from an engineering design textbook by Budynas and Nisbett (2011). Euler buckling theory results in a critical buckling load given as

\[ F_{\text{buckle}} = \frac{C \pi^2 EI}{L_{\text{pan}}^2} \]  

(38)

where \( C \) is the end condition constant, \( E \) is the modulus of elasticity, \( I \) is the second moment of area, and \( L_{\text{pan}} \) is the length of the panel. This approach assumes that the panel has a slenderness ratio above the critical value. The slenderness ratio is given as

\[ \left( \frac{L}{k_{gyr}} \right)_{\text{actual}} = \frac{L_{\text{pan}}}{\sqrt{I/A}} = \frac{\sqrt{12}L_{\text{pan}}}{t_{\text{pan}}} \]  

(39)

where \( k_{gyr} \) is the radius of gyration, \( A \) is the cross sectional area of the panel, and \( t_{\text{pan}} \) is the thickness of the panel. This value can then be compared to the critical slenderness ratio given as

\[ \left( \frac{L_{\text{pan}}}{k_{gyr}} \right)_{\text{cr}} = \sqrt{\frac{2\pi^2 CE}{\sigma_{\text{yield}}}} \]  

(40)

with \( \sigma_{\text{yield}} \) being the yield stress of steel. Note that different end conditions such as both ends fixed or pinned can yield different critical slenderness ratios.
Euler buckling also assumes that there is minimal eccentric loading, or axial loading that is off-center by a small distance called the eccentricity. If eccentric loading is appreciable, the secant column formula must be used for buckling given as

\[
F_{buckle} = \frac{\sigma_{yield} \ast A}{1 + \left(\frac{e_{load} t_{pan}}{2k_{gyr}}\right) \sec \left(\frac{L_{pan}}{2k_{gyr}}\right) \sqrt{F_{buckle}/AE}}
\]

(41)

with \( e_{load} \) being the loading eccentricity. As the formula is implicit, it must be solved iteratively. The panel gradients for this study were examined to decide whether secant buckling needed to be used.

For this study, the only axial forces on the panel are induced forces from thermal expansion. Therefore, it is convenient to write the Euler buckling force as the temperature change necessary to produce the buckling force, written as

\[
\Delta T_{Euler} = \frac{C\pi^2 t_{pan}^2}{12\alpha L_{pan}^2}
\]

(42)

with \( \alpha \) being the coefficient of thermal expansion. The same can be done for the secant buckling equation yielding

\[
\Delta T_{secant} = \frac{\sigma_{yield}}{E\alpha \left(1 + \left(\frac{e_{load} t_{pan}}{2k_{gyr}}\right) \sec \left(\frac{L_{pan}}{2k_{gyr}}\right) \sqrt{\Delta T\alpha}\right)}.
\]

(43)

As computational buckling analysis was also considered for this study, it is convenient to relate a resulting simulation buckling force to the temperature change necessary to induce that force, given as

\[
\Delta T_{comp} = \frac{F_{buckle}}{w_{pan} t_{pan} E\alpha}
\]

(44)
where $w_{pan}$ is the width of the panel. The width was taken to be unity (1 m) so that the two-dimensional analysis could be analyzed per unit depth. A full derivation of Equation (42), Equation (43), and Equation (44) can be found in Appendix D.

### 1.5.3. Abaqus Theory

Abaqus Standard version 6.14 is a structural solver program that uses the finite element method. Abaqus was used for all of the structural simulations in this study. As the panel heats and deforms at a much slower time scale than the fluid reacts for hypersonic flow, the panel was considered quasi-static in transient simulations. In addition, the large deformations of the panel necessitated the inclusion of non-linear effects in the solver. Abaqus’ user guide (2014) recommended that Newton’s method be used for non-linear simulations. For pure buckling, Abaqus solves a traditional buckling eigenvalue problem.

### 1.6 Proper Orthogonal Decomposition

CFD and FEA are useful tools for solving flow fields and thermal responses, but computational costs remain high and modeling turbulence has always been difficult. Work originally by Lumley (1967) and extended by Sirovich (1987) offers an alternative solution to these issues. A paper by Caraballo et al. (2007) utilized their techniques to examine cavity flow. Traditionally, statistical approaches have been use to estimate turbulence, but in the process ignored the internal structures present in turbulent flow. Caraballo et al. proposed using a proper orthogonal decomposition snapshot method to calculate turbulence for cavity flow instead. This technique is computationally inexpensive, and allows parameters to easily be altered and their effects examined. This method involves using inner products to form basis functions from existing experimental or CFD datasets and using the basis functions to develop POD modes. An example inner product for velocity is defined for two vectors by Caraballo as

$$
\langle \bar{u}_1, \bar{u}_2 \rangle_{\alpha} |_S = \int_S \left( u_1 u_2 + v_1 v_2 + \frac{2 \alpha_{con}}{\gamma - 1} c_1 c_2 \right) d\bar{x}
$$

(45)

where $S$ is the flow domain, $\gamma$ is the specific heat ratio, $\alpha_{con} = 1$ is a constant, and the velocity vector defined as $\bar{u}(\bar{x}, t) = [u(\bar{x}, t), v(\bar{x}, t), c(\bar{x}, t)]$ with $u$ being the horizontal velocity, $v$ being
the vertical velocity, and $c$ being the local speed of sound. Since the simulation is two-dimensional, the span wise velocity $w$ is taken to be zero. These inner products are formed from snapshots of the spatial characteristics of the flow at different points in time. These solution sets manifest themselves as scalar or vector fields of data in the spatial dimensions $x$ and $y$, and the temporal dimension $t$. Unfortunately, these snapshots must come from preexisting solution sets which limits the usefulness of the ROM. Special care must also be taken to sample a large number of snapshots in time to improve the accuracy of the POD model. Quantities include velocity, density, pressure, and temperature from points in the flow field. A code to obtain the reduced order models based on the POD approach described above has previously been written by Caraballo et al., and is ready to be tested with hypersonic data. Further explanation can be found in Appendix E.
Chapter 2: Research Methodology

Several simulation scenarios have been examined for this study. The first set is steady state fluid simulations with the goal of validating against a MATLAB model created for this study and between different CFD solver models. The second set examined the panel’s response to thermal heating. The final set examined loosely coupling the fluid-thermal and structural simulations.

2.1 Domain and Mesh Creation

The flow geometry and domain of interest is a panel fixed to supports at each end with hypersonic flow over the top surface. The dimensions and flow parameters were created to emulate the setup used by Thornton and Dechaumphai (1988) which consisted of a flat steel panel affixed at each end to a conductive wall. Air entered fully developed at Mach 4.24 (1443 m/s at standard air conditions) at 295 K. The mesh had quadrilateral elements, extended a quarter of the panel’s length in front of and behind the panel, and extended almost half of the panel’s length vertically. Although not explicitly stated, the support walls seem to have had an isothermal boundary condition based upon the surface temperature plots showing a near constant temperature above the support walls. The top surface of the panel had a thermal interaction with the airflow and radiation to space, the sides conducting to the panel supports, and the bottom surface being perfectly insulated.

For this study, two different meshes A and B were used with quadrilateral elements for both. The meshes were also seeded so that finer resolution would be provided near the boundary layer and the panel, placing approximately ten layers within the velocity boundary layer to emulate Thornton and Dechaumphai (1988). This criteria was met for the both the turbulent and laminar solvers. However, analysis in ANSYS Fluent yielded $y_+ = 17.3$ as the average value along the panel for the $k - \varepsilon$ model. Since this value is greater than $y_+ = 10$, ANSYS Fluent likely used wall functions for turbulent model analysis.

For steady state validation simulations, mesh A was extended significantly from the original study’s mesh with five times the panel length before the panel and above the panel, and ten times the panel length behind the panel. These large distances were designed to prevent boundary conditions from interfering with flow characteristics near the pane. The longer entrance length allows the initially constant velocity profile at the inlet to fully develop by the time it reaches the panel. The inlet was given a constant velocity boundary condition of 1443 m/s with
atmospheric pressure. The outlet and far field boundary were both given far field boundary conditions where a Mach number of 4.24 and atmospheric pressure were specified. The panel was not itself modeled, but partitioned from the rest of the wall. Depending on the scenario, the panel was also given a deformation. A schematic for the setup of mesh A can be seen below in Figure 2, while the mesh can be seen in Figure 27 in Appendix B.

![Mesh A schematic with boundary conditions and dimensions.](image)

**Figure 2:** Mesh A schematic with boundary conditions and dimensions.

**Table 1:** Boundary conditions for mesh A.

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Boundary Condition</th>
<th>Variable</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Velocity Inlet</td>
<td>Velocity</td>
<td>1443</td>
<td>m/s</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure Far field</td>
<td>Gauge Pressure</td>
<td>0</td>
<td>Pa</td>
</tr>
<tr>
<td>Far field</td>
<td>Pressure Far field</td>
<td>Gauge Pressure</td>
<td>0</td>
<td>Pa</td>
</tr>
<tr>
<td>Upstream Wall</td>
<td>Adiabatic Wall</td>
<td>Temperature</td>
<td>295</td>
<td>K</td>
</tr>
<tr>
<td>Downstream Wall</td>
<td>Adiabatic Wall</td>
<td>Temperature</td>
<td>295</td>
<td>K</td>
</tr>
<tr>
<td>Panel</td>
<td>Adiabatic Wall</td>
<td>Heat Flux</td>
<td>0</td>
<td>W/m²</td>
</tr>
</tbody>
</table>
Mesh B closely matches the one used by Thornton and Dechaumphai (1988) to allow direct comparisons to be made. As in the original study, the flow will enter the inlet fully developed, negating the need for a longer entrance wall to allow the flow to develop. The inlet velocity profile was determined by running a preliminary steady state solution on mesh B and extracting the velocity data along a vertical line positioned 0.1397 m from the entrance. A cubic equation was then fit to the data points lying within the boundary layer and can be found in Appendix D. The equation was then input into a UDF for the boundary layer profile with the remaining flow being initialized at the freestream velocity of 1443 m/s.

The panel support walls have an isothermal boundary condition. The panel was fully modeled with the top panel surface having a thermal structural-fluid interaction with the air. Thornton and Dechaumphai (1988) state that the panel sides thermally conduct to the panel supports, but since they do not include any dimensions for the panel supports, the supports were assumed to be thermally massive compared to the panel. Under this assumption, the sides of the panel are given isothermal conditions to approximate semi-infinite conduction into the panel supports. The bottom of the panel is given a sink radiation boundary condition to an infinite sink at \( T = 1 \, K \) to model radiation to space. Thornton and Dechaumphai (1988) model this radiation from the top of the panel, but ANSYS Fluent does not allow sink radiation on coupled surfaces. As thermal gradients through the thickness of the panel are assumed to be negligible, this change in boundary conditions should closely approximate, but not replicate those used by Thornton and Dechaumphai (1988). Details of the setup can be seen below in Figure 3 and in Table 2. The mesh itself can be seen in Figure 27 in Appendix B.
Figure 3: Mesh B schematic with boundary conditions and dimensions.

Table 2: Boundary conditions for mesh B.

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Boundary Condition</th>
<th>Variable</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Velocity Inlet</td>
<td>Velocity</td>
<td>Varies</td>
<td>m/s</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure Far field</td>
<td>Gauge Pressure</td>
<td>0</td>
<td>Pa</td>
</tr>
<tr>
<td>Far field</td>
<td>Pressure Far field</td>
<td>Gauge Pressure</td>
<td>0</td>
<td>Pa</td>
</tr>
<tr>
<td>Upstream Wall</td>
<td>Isothermal Wall</td>
<td>Temperature</td>
<td>295</td>
<td>K</td>
</tr>
<tr>
<td>Downstream Wall</td>
<td>Isothermal Wall</td>
<td>Temperature</td>
<td>295</td>
<td>K</td>
</tr>
<tr>
<td>Panel Top Boundary</td>
<td>Fluid/Structural</td>
<td>None</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>Panel Side Walls</td>
<td>Isothermal</td>
<td>Temperature</td>
<td>295</td>
<td>K</td>
</tr>
<tr>
<td>Panel Bottom Wall</td>
<td>Sink Radiation</td>
<td>Temperature</td>
<td>1</td>
<td>K</td>
</tr>
</tbody>
</table>

A mesh convergence study was also done on Mesh B to ensure that the physics of the system would be properly captured. A 5 s simulation was run on five different meshes with different numbers of elements. The boundary layer thickness at the center of the plate and the average top panel surface temperature were monitored to analyze the mesh convergence. The boundary layer thickness seems to have nicely converged to a value of around 0.37 mm. Convergence is less definitive when examining the average temperature of the top panel surface,
but still seems to be converged to a value of about 340 \( K \). Therefore, confidence is felt that mesh B accurately captures the physics of the problem. The results can be seen below in Figure 4.

![Figure 4: Plots of mesh convergence for flow on mesh B after 5 s for (A) boundary layer thickness and (B) average top panel surface temperature.](image)

In the study by Thornton and Dechaumphai (1988), the panel is made from AM-350 stainless steel with temperature dependent properties. This study duplicates those parameters defining the temperature dependent properties of thermal conductivity, specific heat, the thermal expansion coefficient, and the modulus of elasticity for steel. The properties of AM-350 stainless steel are referenced from the database of the steel company High Temp Metals (2015) and can be seen in Table 15 and Table 16 in Appendix H.

The properties of air are matched as closely as possible to those used in the study by Thornton and Dechaumphai (1988) using temperature dependent formulations found in ANSYS Fluent. The ideal gas law formulation is used for density, the kinetic theory formulation for thermal conductivity, and the Sutherland formulation for viscosity. A piece-wise linear interpolation of empirical data method is used for the specific heat of air at various temperatures based on values from a thermodynamics textbook by Cengel and Boles (2002). These formulations used for air are valid for a range of at least \( T = 265 \ K \) to \( T = 1,000 \ K \) and a summary of them can be seen in Table 3 below.
### Table 3: Temperature dependent properties of air.

<table>
<thead>
<tr>
<th>Property</th>
<th>Method</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specific Heat</td>
<td>Empirical Data Interpolation</td>
</tr>
<tr>
<td>Density Formulation</td>
<td>Ideal Gas Law</td>
</tr>
<tr>
<td>Viscosity Formulation</td>
<td>Sutherland Formulation</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>Prandtl Number Definition</td>
</tr>
</tbody>
</table>

#### 2.2 MATLAB Model

The supersonic oblique shock equations were used for validation of the ANSYS Fluent model. As these equations are rather complicated and even involve implicit iteration, a MATLAB model was developed to solve Equations (19) through (37). Initial parameters of Mach number, pressure, density, and temperature are given along with $x$ and $y$ coordinate vectors for a two-dimensional geometry. The model then examines one section of the geometry at a time and determines the turn angle $\theta$. If the angle is positive, the shock wave equations are used to determine the shock angle $\beta$ along with the values of the state variables across the shock wave. If the angle is negative, the Prandtl-Meyer function is used to determine the values for the variables for the expansion wave including the Mach angles. After the model has calculated the state variables for all of the sections, a plot of the geometry with the shock waves and expansion waves is created. The waves are at the correct angles and with a length proportional to the magnitude of the pressure ratio across that wave. The Mach number, the pressure, the density, and the temperature is output as data for comparison with the ANSYS Fluent model. A flow diagram of the MATLAB program can be found in Figure 26 in Appendix A. The inputs and outputs of the MATLAB model are summarized below in Table 4.
Table 4: Inputs and outputs of the MATLAB model.

<table>
<thead>
<tr>
<th>Freestream Inlet Inputs</th>
<th>Outputs per Geometry Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-coordinates of geometry</td>
<td>Mach number</td>
</tr>
<tr>
<td>Y-coordinates of geometry</td>
<td>Pressure</td>
</tr>
<tr>
<td>Mach number</td>
<td>Temperature</td>
</tr>
<tr>
<td>Pressure</td>
<td>Density</td>
</tr>
<tr>
<td>Temperature</td>
<td>Static pressure</td>
</tr>
<tr>
<td>Density</td>
<td>Static density</td>
</tr>
<tr>
<td>Specific heat ratio</td>
<td>Static density</td>
</tr>
<tr>
<td></td>
<td>Shock angle or Mach angles</td>
</tr>
</tbody>
</table>

An example output from the MATLAB model can be seen below in Figure 5. The shock waves are shown as long dashed lines. The expansion waves are shown as dashed-dotted lines with the forward and rearward Mach lines displayed. Note that the vertical and horizontal axes were scaled differently for clarity, and therefore the shock wave, expansion wave, and geometry angles are not displayed accurately for this example.
Figure 5: Sample output from MATLAB model showing shock waves and expansion waves over a trapezoidal geometry.

2.3 Steady State Simulations

The first simulation approach assumes an unchanging panel shape for several geometries and examines the steady state fluid response. This approach has the advantage of simplicity and being able to be compared to the theoretical oblique shock equations. Simple flat plate and wedge geometries were run first as they can directly be compared to the oblique shock and Prandtl-Meyer equations. The MATLAB model constructed can easily solve the equations for more complicated systems with multiple compression and expansion corners that are difficult to solve manually. Simulations were also run for smoother geometries such as a convex arc and a concave arc that are more typically found in panel buckling. These were examined in MATLAB by approximating the smooth arcs with linear splines, but caution was applied when comparing the solution to ANSYS Fluent results, as the oblique shock equations were derived assuming discrete angle geometries. Curvature may cause inaccuracies in the solution such as non-isentropic conditions around expansion corners that must be considered.
Four basic steady state geometries were considered: a convex arc emulating a deformed panel, a concave arc, a convex trapezoid approximating the convex arc, and a concave trapezoid approximating the concave arc. A flat panel was also included as a base case to ensure that the simulations were running correctly, but no results were included. No panel thickness was modeled in these simulations. The arcs attempt to mimic actual panel deformation and are defined by three points: the left and right end of the panel, and a third point centered at the height of the specified deflection. The trapezoids are included to provide applicable geometry for use with the MATLAB model as it assumes discrete angles. The trapezoids are intended to approximate the shape of the arcs. Their height is simply the deflection. To define the angle of the corners, a geometric equation is used which causes the legs of the trapezoid to intersect the arc at exactly half of the deflection height. The corner angle is defined below as

$$\theta = \tan^{-1}\left(\frac{2d}{L_{pan} \cdot (2 - \sqrt{3})}\right)$$

where \(d\) is the deflection height and \(L_{pan}\) is the panel length. A derivation of this expression is available in Appendix D. A visual representation of the five geometries can be seen below in Figure 6.

Note: Deflections are exaggerated in this diagram for clarity

**Figure 6:** The five geometric setups for steady state simulations.
Another consideration when running CFD simulations is solution convergence. Since the Navier-Stokes equations are being solved iteratively, care must be taken to assure enough iterations were run for an accurate solution to be found. Various convergence monitors were used for this study to ensure convergence including scaled residuals, mass flow rate imbalance, gross mass flow rate, and mass-averaged outlet temperature, velocity, pressure and density. These monitors allowed convergence to be determined by monitoring the behavior of these parameters over time.

2.4 Structural Simulations

In the second simulation approach, Abaqus was used to perform structural simulations of the panel under various thermal loads to compare to the data given by Thornton. Two different setups were used emulating the models used by Thornton and Dechaumphai (1988) with one inducing convex deformation upward, and the other inducing concave deformation downward. These will be referred to as the convex and concave setups respectively. For the convex setup, the bottom two corners of the panel were given pin boundary conditions preventing translation, but allowing rotation. Similarly, the concave setup has pin boundary conditions on the top two corners. These setups can be seen below in Figure 7.

![Figure 7: Boundary conditions for elastic deformation structural analysis.](image)

The panels were given initial conditions of no stress and a constant temperature of 295 K. Different temperature loads were then applied to examine the resulting maximum deflection and maximum von Mises stress.
Unfortunately, the boundary conditions used above on the panel are not fully realistic, and were chosen partially for their ability to induce exclusively convex or concave deformation. Simple buckling analysis performed in Abaqus examines the validity of these convex and concave panel setups. For the buckling analysis, both ends pinned and both ends fixed boundary conditions were used as these would be more typical in an actual hypersonic aircraft. Both studies led by Ko (1995; 2004) feature these boundary conditions for panels. The geometries can be seen below in Figure 8.

![Figure 8: Boundary conditions for structural buckling analysis.](image)

Euler buckling theory allows simple estimates of buckling forces to be made, but the slenderness ratio condition must be met. Using the dimensions of the panel and the temperature properties of AM-350 stainless steel at standard air conditions, the slenderness ratio was found by Equation (39) to be

\[
\frac{L}{k_{gyr}}_{\text{actual}} = \frac{\sqrt{12L}}{t_{pan}} = \frac{\sqrt{12}(0.1016 \text{ m})}{(0.00254 \text{ m})} = 138.56
\]

Next, the critical slenderness ratio for pinned and fixed end condition were found using Equation (40) to be
\[
\left( \frac{L}{k_{gyr}} \right)_{cr, pin} = \sqrt{\frac{2\pi^2 CE}{\sigma_{yield}}} = \sqrt{\frac{2\pi^2 (1)(1.88 \times 10^{11} \text{ Pa})}{(9.79 \times 10^8 \text{ Pa})}} = 61.57 < \left( \frac{L}{k_{gyr}} \right)_{actual}
\]

\[
\left( \frac{L}{k_{gyr}} \right)_{cr, fix} = \sqrt{\frac{2\pi^2 CE}{\sigma_{yield}}} = \sqrt{\frac{2\pi^2 (4)(1.88 \times 10^{11} \text{ Pa})}{(9.79 \times 10^8 \text{ Pa})}} = 123.14 < \left( \frac{L}{k_{gyr}} \right)_{actual}
\]

It can be seen that the actual slenderness ratio is higher than the critical value for both end conditions. The slenderness ratio is somewhat close to the critical slenderness ratio for fixed end conditions, but according to Budynas and Nisbett (2011), it is only important that the slenderness ratio be above the critical value. Therefore, Euler buckling theory is appropriate to use for the panel with that criteria.

Euler buckling theory also requires that there be no eccentric loading for pinned conditions. Eccentric loading does not affect beams with fixed conditions due to the fixed ends preventing any initial curvature from occurring. If the temperature difference between the top and the bottom of the panel is too large, the thermal expansion could result in an eccentric load. For this study, it was assumed that such thermal gradients are negligible through the panel’s thickness, but that assumption needed examination. It was found that if a linear temperature profile is assumed between the top and bottom of the panel, the eccentricity caused by differing thermal strains is

\[
e_{load} = \frac{\left( T_0 - \frac{T_{top} + T_{bot}}{2} \right) + \sqrt{\left( \frac{T_{top} + T_{bot}}{2} - T_0 \right)^2 - \left( \frac{T_{top} - T_{bot}}{2} \right)^2}}{\frac{T_{top} - T_{bot}}{t_{pan}}}
\]

(47)

where \( T_{top} \) is the temperature of the top of the panel, \( T_{bot} \) is the temperature of the bottom of the panel, and \( T_0 = 295 \text{ K} \) is the initial temperature of the panel. Derivation of these results can be found in Appendix D. In order to see the effect of eccentricity on buckling behavior, it is assumed that
\[
\Delta T_{\text{pan}} \approx \left( \frac{T_{\text{top}} + T_{\text{bot}}}{2} - T_0 \right)
\]  

(48)

where \( \Delta T_{\text{pan}} \) is the difference between the average panel temperature and the initial temperature. Equation (47) can then be combined with Equation (43) to examine the effects of eccentricity for several different combinations of \( T_{\text{top}} \) and \( T_{\text{bot}} \). Some sample values can be seen below in Table 5. \( \Delta T_{\text{pan}} \) is the actual temperature difference experienced by the panel, \( \Delta T_{\text{Euler}} \) is the temperature difference needed to cause Euler buckling, and \( \Delta T_{\text{ecc}} \) is the modified temperature difference from the results of eccentricity, and Error is the percentage difference between the Euler buckling and eccentric buckling temperature differences. Values are listed in order of increasing \( \Delta T_{\text{pan}} \). Note that the panel would actually buckle under the conditions in a given row if \( \Delta T_{\text{pan}} > \Delta T_{\text{ecc}} \). In addition, since \( \Delta T_{\text{ecc}} \) depends on both \( \Delta T_{\text{pan}} \) and the difference between \( T_{\text{top}} \) and \( T_{\text{bot}} \), its value does not necessarily increase in a linear fashion.

<table>
<thead>
<tr>
<th>( T_{\text{top}} ) (K)</th>
<th>( T_{\text{bot}} ) (K)</th>
<th>( \Delta T_{\text{pan}} ) (K)</th>
<th>( e_{\text{load}} ) (m)</th>
<th>( \Delta T_{\text{ecc}} ) (K)</th>
<th>( \Delta T_{\text{Euler}} ) (K)</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>300</td>
<td>295</td>
<td>2.5</td>
<td>( 1.27 \times 10^{-3} )</td>
<td>31.1</td>
<td>42.8</td>
<td>27.49</td>
</tr>
<tr>
<td>320</td>
<td>300</td>
<td>15.0</td>
<td>( 4.85 \times 10^{-4} )</td>
<td>37.2</td>
<td>42.8</td>
<td>13.21</td>
</tr>
<tr>
<td>340</td>
<td>300</td>
<td>25.0</td>
<td>( 6.35 \times 10^{-4} )</td>
<td>35.8</td>
<td>42.8</td>
<td>16.47</td>
</tr>
<tr>
<td>342</td>
<td>332</td>
<td>42.0</td>
<td>( 7.59 \times 10^{-5} )</td>
<td>41.8</td>
<td>42.8</td>
<td>2.41</td>
</tr>
<tr>
<td>340</td>
<td>335</td>
<td>42.5</td>
<td>( 3.74 \times 10^{-5} )</td>
<td>42.3</td>
<td>42.8</td>
<td>1.20</td>
</tr>
</tbody>
</table>

Table 5: The effect of eccentricity on panel buckling for pinned conditions.

Table 5 shows that loading eccentricity from thermal gradients can cause a large percentage change in the buckling temperature difference as seen in the first three rows. However, the thermal conditions in those three rows would not lead to thermal buckling as \( \Delta T \) is too low. Conversely, the thermal conditions in the last two rows would cause buckling, but only lower the buckling temperature difference by less than 1 K. Several more cases were tested, but not reported, and the thermal conditions in the fourth row were found to yield the lowest buckling temperature difference. Additionally, all of these scenarios except for the one in the first row involve temperature differences through the panel thickness of at least 10 K which is highly unrealistic.
considering the high thermal conductivity of stainless steel. It was then concluded that eccentric loading from thermal gradients would have a negligible effect on the buckling behavior of the panel and Euler buckling could be used for accurate buckling estimation.

Finite element eigenvalue buckling analysis was also performed in Abaqus. An end load of unity (1 \( N \)) was used so that the buckling eigenvalue equaled the buckling force for each analysis. This buckling load was then converted to a temperature change using Equation (44).

### 2.5 Loosely Coupled Simulations

In the third approach, simulations were loosely coupled. Practically speaking, coupling a simulation means that the temperature values from the air are transferred to the top surface of the panel, and the vertical position of the panel is transferred to the air. This can either be done automatically via lockstep coupling, or iteratively manually transferred.

Iterative manual coupling, or loose coupling, is simpler and was used for this study. In the study by Thornton and Dechaumphai (1988), the simulations were 30 s in duration, but consisted of three smaller simulations. The flow field was first transiently for \( 4 \times 10^{-4} \) s using time steps of \( 1 \times 10^{-7} \) s until the flow field reached semi-steady state conditions for an initially flat plate. The time step was then increased to \( 1 \times 10^{-2} \) s to reach 10 s. At that point, the temperature profile was transferred to a separate quasi-static structural simulation to determine the deflection response. The deflection profile was then output to the flow solver. This process was then repeated two more times until the final flow field and structural deflection profile was known.

A similar approach was taken in this study. ANSYS Fluent solved the flow field at the beginning of each 10 s using \( 1 \times 10^{-7} \) s time steps until the flow stabilized. The time steps were then increased to approximately \( 1 \times 10^{-2} \) s to solve the thermal conditions for a full 10 s. The panel surface temperature profile was then output to Abaqus which solved the panel deformation. Initially, the temperatures were manually exported from ANSYS Fluent and manually imported into Abaqus. The resulting maximum panel deflection was then input into a UDF which deformed the panel at the appropriate time using a smooth arc approximation as exporting the deflections from the entire top surface of the panel proved infeasible. The formulation for the smooth arc approximation can be found in Equation (62) in Appendix D. With the deflection profile being updated, the process was then repeated for another 10 s. The procedure is outlined below in Figure 9.
Loosely coupled simulations with different transfer intervals followed a similar procedure, but used different step sizes to adjust for different transfer intervals being considered in each scenario. Regardless of the transfer interval, the flow was allowed to stabilize for a full $4 \times 10^{-4}$ s using 4,000 time steps of $1 \times 10^{-7}$ s each after the UDF undated the deformation.

### 2.6 Sinusoidally Driven Simulations

The last simulation considered was to assume a sinusoidally oscillating panel deformation to examine its transient effect on the flow field. A half-period sine wave implemented in a user-defined function (UDF) could be used to deform the panel to examine flutter experienced in hypersonic flight. In order to run these simulations, knowledge of fluttering frequencies and temperature thresholds would have to be examined.

A UDF for sinusoidal oscillations was developed based off of a UDF from an ANSYS Fluent butterfly valve tutorial by Hiemcke (2004) with a moving membrane. However, it was ultimately decided that flutter was somewhat outside of the scope of this study, and a few sinusoidal simulations were run for proof of concept purposes only. Performing full loosely coupled simulations on flutter behavior is left for future work. The equation used by the sinusoidal UDF can be found in Appendix D.
Chapter 3: Results and Discussion

3.1 Steady State Simulation Results

Several simulations were run using the five different geometries described in Figure 6 from section 2.3 to examine the effects of different turbulence models and material properties.

3.1.1. Comparing Turbulence Models

The first goal was to compare different turbulence models with each other and with models ignoring turbulence including the MATLAB model based upon Equations (19) through (37). The convex and concave trapezoid geometries were chosen for this task with a deflection of $\frac{8.382 \times 10^{-4}}{m}$. The turbulence models considered were the Spalart-Allmaras, $k - \epsilon$, and $k - \omega$ models, in addition to the non-turbulent inviscid, laminar, and MATLAB models. In order to ensure appropriate comparison, adiabatic wall boundary conditions and constant specific heat definitions were used in the four non-flat geometries to match the assumptions made in the derivation of the MATLAB model. The inviscid model was expected to match most closely with the MATLAB model as both ignore viscous dissipation and turbulence effects. The Spalart-Allmaras, $k - \epsilon$, and $k - \omega$ models were expected to closely match each other as all take into account viscous dissipation along with modeling turbulence. The results from the laminar model were expected to differ slightly from both of these groups as it accounts for viscous dissipation, but ignores turbulence effects.

Comparisons were made using the absolute pressure data taken in a horizontal line across the domain. For geometries with positive deformations, the line was positioned at 10% of the deformation height above the deformation ($y = 9.220 \times 10^{-4} m$). For geometries with negative deformations, the line was likewise positioned at 10% of the deformation magnitude above the bottom of the domain ($y = 8.382 \times 10^{-5} m$). A plot of the absolute pressure versus the distance can be seen below in Figure 10 for the convex trapezoidal geometry. An unscaled diagram of the geometry has been included below the plot in Figure 10 and in following figures for clarity.
Figure 10: Plot of absolute pressure vs position for MATLAB model and different ANSYS Fluent models on the convex trapezoidal geometry at $M = 4.24$.

The pressure maximum in Figure 10 corresponds to the initial compression and expansion corners while the minimum corresponds to the final expansion and compression corners. As expected, the inviscid model most closely matches the MATLAB model and the other three turbulence models agree well with each other. However, the pressure jumps in the three viscous, turbulence models were somewhat lower than those in the inviscid and MATLAB models. Interestingly, the laminar solver, though viscous, most closely matches the inviscid and MATLAB model. This most likely is due to the laminar solver failing to account for energy removed from the flow in the form of turbulent energy.

Similar trends can be seen below in Figure 11 for the concave trapezoidal geometry.
Figure 11: Plot of absolute pressure vs position for MATLAB model and different ANSYS Fluent models on the concave trapezoidal geometry at $M = 4.24$.

As the three turbulence models tested are applicable for hypersonic flow and all agreed well with each other, confidence was felt in using any of them for further study. The $k - \epsilon$ model was chosen for the rest of the simulations as the ANSYS Fluent user guide (2013) labels it as a robust solver and Roy and Blotter (2006) state the $k - \epsilon$ model has gone through extensive validation over the years. While not examined in as much detail, the $k - \omega$ model is similarly well validated and was used in one simulation to check the results of the $k - \epsilon$ model.

The convex and concave arc geometries were examined next using the MATLAB model, the inviscid model, the laminar model, and the $k - \epsilon$ model. In order to use the MATLAB model for the curved geometries, curves had to be approximated as linear splines. Initially, the arc was estimated using lines connecting 100 points. As expected, the MATLAB model gave inaccurate results as the oblique shock equations were not derived with curved surfaces in mind. The maximum pressure rise predicted by the MATLAB model was nearly twice that predicted by any of the ANSYS Fluent models which are virtually indistinguishable at the scale shown. The issue
was traced to the linear approximation of the arc. In the trapezoidal simulations, the initial turn angle on the geometry was $3.526^\circ$ while with the arc simulations approximated by 100 points that initial angle jumped to $9.277^\circ$. The ANSYS Fluent models correctly accounted for the angular jump by taking into account the overall curvature of the geometry, but the MATLAB model only considered the magnitude of the initial turn angle and therefore drastically over predicted the pressure rise. In particular, the MATLAB model’s predictions for the shock waves are worse than those for the expansion waves. The results of the simulations for the convex arc geometry can be seen below in Figure 12.

![Figure 12: Plot of absolute pressure vs position for MATLAB model (100 points) and different ANSYS Fluent models on the convex arc geometry at $M = 4.24$.](image)

Due to the large inaccuracies of the results, a study was done on the number of points used to approximate the curve for the convex arc geometry. It was found that fewer points used in the linear approximation yielded greater accuracy in predicting the maximum and minimum pressure values. While slightly counter-intuitive, the increased accuracy from using fewer points stems from the initial turn angle being lower. In order to use the MATLAB model for curved surfaces,
the number of points must be tuned to give as accurate values of the pressure magnitudes as possible. Due to this number of points based accuracy, the MATLAB model results are not considered definitive for any simulations run on the convex and concave arc surfaces. For this study, five points were chosen based upon the results can be seen below in Figure 13 knowing that the MATLAB model should approximately match the inviscid results in Figure 10 and Figure 11. Aside from Figure 12 and Figure 13, all MATLAB results in this study were calculated using five points.

**Figure 13:** Plot of absolute pressure vs position for MATLAB model using various numbers of points and ANSYS Fluent inviscid model on the convex arc geometry at $M = 4.24$.

### 3.1.2. Comparing Velocities

All the simulations thus far considered were run at Mach 4.24. For a more thorough and realistic model validation, different flow velocities were also considered. Five flow speeds were chosen ($M = 3.24, 3.74, 4.24, 4.74,$ and $5.24$) and run using the MATLAB model, the inviscid model, the laminar model, and the $k - \epsilon$ model on the four non-flat geometries. The pressure maximums and minimums from the horizontal line sampled in the domain were then plotted versus
the corresponding Mach number. The results can be seen below in Figure 14, Figure 15, Figure 16, and Figure 17.

**Figure 14:** Plot of absolute pressure maximums and minimums vs Mach number for MATLAB model (5 points) and different ANSYS Fluent models on the convex trapezoidal geometry.
**Figure 15**: Plot of absolute pressure maximums and minimums vs Mach number for MATLAB model (5 points) and different ANSYS Fluent models on the concave trapezoidal geometry.

**Figure 16**: Plot of absolute pressure maximums and minimums vs Mach number for MATLAB model (5 points) and different ANSYS Fluent models on the convex arc geometry.
The figures above show that the MATLAB and inviscid models agree well over a broad range of velocities for the trapezoidal geometries which is especially encouraging considering the curvature issues inherent to the MATLAB model. The $k - \epsilon$ model consistently yields maximum absolute pressures slightly below the other two models’ maximums and minimums absolute pressures slightly above the other models’ minimums. This is likely due to the conversion of more energy into temperature gradients and turbulent kinetic energy in the $k - \epsilon$ model due to turbulence. The additional heat transfer from the turbulent thermal conductivity portion of $k_{eff}$ in Equation (3) most likely also has a large effect on increasing the overall heat transfer. The results for the laminar solver were less conclusive. Theoretically, the laminar model should have yielded pressure maximums and minimums between the MATLAB and inviscid models, and the $k - \epsilon$ model by accounting for the pressure energy lost to viscous dissipation, but failing to account for the turbulent kinetic energy. This trend seemed true for the pressure minimums at higher velocity, but otherwise was inconsistent. The exact reason for this behavior is unknown, but could possibly be ascribed to variations arising from the sampling location. Since all of the pressure readings were taken at one specific height in the domain, it is possible that differences in the location and the
changing angles of the shock and expansion waves caused pressure extreme locations to shift off the sampling line. If this were the case, it is not known why the laminar model seems to be the only model affected.

Ultimately, the different models show good agreement over a range of supersonic and hypersonic velocities giving confidence in their use in further simulations.

### 3.1.3. Temperature Independent versus Dependent Material Properties

Simulations were also run comparing air defined with constant properties and with the temperature dependent properties described in section 1.4.1 using the laminar and $k - \epsilon$ models at $M = 4.24$ with a convex panel deformation. Pressure distributions were sampled along a horizontal line 10% higher ($8.382 \times 10^{-5} m$) than the top of the deformation. Temperature distributions were sampled along a vertical line in the flow at the midpoint of the panel ($x = 0 m$) from the surface to 0.010 m in height to capture the thermal boundary layer. The results can be seen below in Figure 18 and Figure 19.

Figure 18: Plot of absolute pressure vs horizontal position for the ANSYS Fluent laminar and $k - \epsilon$ models using constant and temperature dependent properties on the convex trapezoidal geometry at $M = 4.24$. 

---

**Figure 18:** Plot of absolute pressure vs horizontal position for the ANSYS Fluent laminar and $k - \epsilon$ models using constant and temperature dependent properties on the convex trapezoidal geometry at $M = 4.24$. 

---
Figure 19: Plot of temperature vs height above the plate for the ANSYS Fluent laminar and $k-\epsilon$ models using constant and temperature dependent properties on the convex trapezoidal geometry at $M = 4.24$.

For the laminar model, it can be seen that the use of temperature dependent properties led to smaller pressure spikes, while the same properties had little discernible effect on the pressure of the $k-\epsilon$ model. Temperature dependent properties also cause a mild increase in the temperature for the $k-\epsilon$ model overall, while only causing a similar temperature rise in the laminar model very near to the surface. It then would seem likely that using constant material properties versus temperature dependent ones in hypersonic flow at temperatures below 1,000 K causes slight, but meaningful errors. The inclusion of temperature dependent properties in the coupled simulations therefore seems prudent, and necessary in matching the results from Thornton and Dechaumphai (1988).

A more discernible trend can be seen between the laminar and $k-\epsilon$ model where the laminar model appears to predict much lower viscous dissipation and heat transfer in the flow. This is a logical conclusion as the $k-\epsilon$ model introduces an extra turbulent viscous term in the form of Reynolds stresses in addition to the extra thermal conductivity from turbulence that the laminar model neglects.
Taken together, all of these steady state results possibly indicate that usage of a laminar solver may not have been entirely valid as pressure gradients could have been exaggerated and temperature underrepresented, especially in the boundary layer. However, to reiterate section 1.4, as neither the initial Reynolds number nor the wall length prior to the inlet were known from Thornton and Dechaumphai (1988), the judgment of the validity of the laminar solver is partially speculative. In addition, while using temperature dependent properties instead of constant properties for air does not seem to cause large changes in results, it ensures that the simulations are as accurate as possible. The effect of temperature dependent properties may also be even more pronounced with transient simulations.

3.2 Structural Simulation Results

Structural simulations were run to analyze the behavior of the panel under thermal loading for convex, concave, pinned, and fixed panel setups.

3.2.1. Elastic Deformation

Abaqus was used to thermally analyze the convex and concave panel setups found in the study by Thornton and Dechaumphai (1988). That study gave the deflection and temperature distribution for the convex panel setup at t = 0, 10, 20, and 30 s. For this study, a two-dimensional panel was given the same approximate temperature load using a constant average value, and its maximum deflection and maximum stress recorded. The results can be seen below in Table 6 and Table 7.

Table 6: Thornton and Abaqus panel deflections for various average temperatures through time.

<table>
<thead>
<tr>
<th>Time (s)</th>
<th>Average Temperature (K)</th>
<th>Deflection: Thornton (m)</th>
<th>Deflection: Abaqus (m)</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>295.0</td>
<td>0</td>
<td>0</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>332.2</td>
<td>3.30 × 10^{-4}</td>
<td>3.09 × 10^{-4}</td>
<td>6.40</td>
</tr>
<tr>
<td>20</td>
<td>361.1</td>
<td>6.10 × 10^{-4}</td>
<td>5.49 × 10^{-4}</td>
<td>9.96</td>
</tr>
<tr>
<td>30</td>
<td>391.7</td>
<td>8.38 × 10^{-4}</td>
<td>8.27 × 10^{-4}</td>
<td>1.37</td>
</tr>
</tbody>
</table>
Table 7: Thornton and Abaqus average top panel stresses for various average temperatures through time.

<table>
<thead>
<tr>
<th>Time (s)</th>
<th>Average Temperature (K)</th>
<th>Average Stress: Thornton (Pa)</th>
<th>Average Stress: Abaqus (Pa)</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>295.0</td>
<td>0</td>
<td>0</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>332.2</td>
<td>$4.14 \times 10^7$</td>
<td>$4.04 \times 10^7$</td>
<td>2.40</td>
</tr>
<tr>
<td>20</td>
<td>361.1</td>
<td>$8.62 \times 10^7$</td>
<td>$7.08 \times 10^7$</td>
<td>17.81</td>
</tr>
<tr>
<td>30</td>
<td>391.7</td>
<td>$1.21 \times 10^8$</td>
<td>$1.05 \times 10^8$</td>
<td>12.74</td>
</tr>
</tbody>
</table>

The results are similar for the two methods, but the stress values do not match as closely in the higher temperature range. One known difference between the two methods is in the material properties of AM-350 steel. The study by Thornton and Dechaumphai (1988) stated that temperature dependent properties were used for properties such as thermal conductivity and specific heat, but no empirical data or formulations were given. For this study, empirical temperature dependent properties were used for the steel. Any differences from the values or formulations used by Thornton and Dechaumphai (1988) could introduce variation. In particular, material property differences in the coefficient of thermal expansion and the modulus of elasticity would influence the deflection and stress results.

Thornton and Dechaumphai (1988) also considered plasticity effects while this study only considered elastic deformation. Elastic deformation is only a valid assumption if the von Mises stresses in the plate are below the yield stress of the panel. For the largest deformation at $t = 30$ s for the pinned convex boundary conditions, the maximum von Mises stress (neglecting the artificial spike found at the pinned nodes resulting from a finite mesh) was $8.17 \times 10^8$ Pa, which is below the yield stress of $9.79 \times 10^8$ Pa. Data can be found in Appendix G. Therefore, elastic deformation seems to be a valid explanation for this scenario, but even small differences from the deformation model used by Thornton and Dechaumphai (1988) could introduce error.

Additionally, reading single values from the temperature plots in Thornton and Dechaumphai (1988) was difficult as the temperature distributions were not constant across the plate. An average value had to be taken for each time step and offers a final source of potential error.
3.2.2. Buckling

In order to try to capture more realistic panel deformation behavior, basic two-dimensional buckling analysis was carried out for the first three modes in Abaqus for the panel with the pinned and fixed boundary conditions from Figure 8. The buckling forces were then converted to equivalent temperature changes using Equation (44). The results can be seen below in Table 8.

Table 8: Buckling forces and temperatures from Abaqus FEA for pinned and fixed support boundary conditions.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Pinned Supports</th>
<th>Fixed Supports</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Force (N)</td>
<td>ΔT (K)</td>
</tr>
<tr>
<td>1</td>
<td>2.69 × 10^5</td>
<td>47.0</td>
</tr>
<tr>
<td>2</td>
<td>1.16 × 10^6</td>
<td>202.3</td>
</tr>
<tr>
<td>3</td>
<td>2.570 × 10^6</td>
<td>448.9</td>
</tr>
</tbody>
</table>

The results indicate that the lowest mode is the most likely to occur due to the low temperature change involved as compared to the higher modes. The first mode buckling temperature changes for the two boundary conditions were then compared to those predicted by the Euler buckling theory in Equation (42). The results can be seen below in Table 9.

Table 9: Temperature changes to cause buckling from Abaqus FEA and Euler buckling theory.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Pinned Supports</th>
<th>Fixed Supports</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Abaqus (K)</td>
<td>Euler (K)</td>
</tr>
<tr>
<td>1</td>
<td>47.0</td>
<td>42.8</td>
</tr>
</tbody>
</table>

The results between the Abaqus and Euler buckling techniques agree fairly well to within a few degrees giving credibility to the technique. The Abaqus results were used in a UDF to better predict panel behavior in coupled simulations.
3.3 Loosely Coupled Simulation Results

Loosely coupled simulations combined the fluid-thermal and structural solvers. Data were saved periodically for several simulations to be used for a later study on ROM.

3.3.1. Comparison with Results from Thornton and Dechaumphai (1988)

For the main part of this study, fluid and structural simulations were loosely coupled to compare with the results of Thornton and Dechaumphai (1988). To ensure a fair comparison, the process had to be modified slightly. Like Thornton and Dechaumphai (1988), the flow field was allowed to stabilize and reach essentially steady state conditions by running the simulation for \(4 \times 10^{-4}\) s using 4,000 time steps of \(1 \times 10^{-7}\) s each. The small timescale was necessary while the flow field was in transient conditions. At this point, the study by Thornton and Dechaumphai (1988) was run using only the thermal solver for 10 s with a time step of \(1 \times 10^{-2}\) s. However, the capabilities of ANSYS Fluent would not allow this to be done. Instead, the time step was changed to \(9.9996 \times 10^{-3}\) s seconds and run for 1,000 time steps to complete the combined fluid-thermal analysis to 10 s. Since the panel temperature increased only slightly between each time step, it was assumed that the flow did not require the small timescale as it was approximately steady the entire time. After 10 s, the temperature profile of the top of the panel was exported to Abaqus and a quasi-static structural simulation was completed. The maximum deflection was then input into a UDF which would appropriately update the panel.

At this point the flow field was stabilized again for another \(4 \times 10^{-4}\) s to account for the change in geometry, and then run for 1,000 larger time steps to reach \(t = 20\) s. The process was then completed again to complete a full 30 s simulation. The temperature and heat flux results of this simulation for the convex panel boundary conditions can be seen below in Figure 21. These can be compared to the temperature and heat flux results from Thornton and Dechaumphai (1988) in Figure 20. The units of the results from the simulation have been changed to match those used by Thornton and Dechaumphai (1988). The same plots in SI units can be found in Figure 28 and Figure 29 in Appendix C. Pressure field plots from the simulation can likewise be found in Figure 30 in Appendix C.
Figure 20: Plot of (A) the top panel surface temperature profile vs position and of (B) the top panel surface heat flux profile vs position from Thornton and Dechaumphai (1988).

Figure 21: Plot of (A) the top panel surface temperature profile vs position and of (B) the top panel surface heat flux profile vs position for ANSYS Fluent simulations.

These above figures show that the temperature plot of loosely coupled simulation match the trends from Thornton and Dechaumphai (1988) fairly well. The panel appears to have a uniform temperature distribution, but with a slight slope opposite of that seen in Figure 20 (A).
bump in the temperature magnitude can be seen at the leading edge of the panel at \( t = 30 \text{ s} \), most likely due to a large shock wave. The temperature profiles for \( t = 10 \text{ s} \) and \( t = 20 \text{ s} \) interestingly do not display this at all with a slightly lower value at the beginning. However, the heat flux plots show two different trends. Large spikes are seen in the heat flux at the ends of the panel in Figure 21, but are not present in Figure 20. Such behavior is deemed a side effect of the boundary conditions used in the panel. The air in the boundary layer above the panel is hot and only increases in temperature as the panel heats up as well over time. However, the isothermal boundary condition used on the ends of the panel creates a large temperature gradient with the air at that location leading to high heat fluxes. The same phenomena is likely not seen in the study by Thornton and Dechaumphai (1988) due the ends of the panel having finite conductive heat flux to the panel holders. In addition, the average temperature and average heat flux at each time increment is noticeably higher in the simulation, especially after the full 30 s time period. The comparison of the average temperatures, average heat fluxes, and maximum deflections can be seen below in Table 10, Table 11, and Table 12.

**Table 10:** Temperature averages for the top panel surface at time intervals of \( t = 0, 10, 20, \) and \( 30 \text{ s} \) for Thornton and ANSYS Fluent laminar solver with convex boundary conditions.

<table>
<thead>
<tr>
<th>Time (s)</th>
<th>Thornton Avg. Temp. (K)</th>
<th>Fluent Avg. Temp. (K)</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>295</td>
<td>295</td>
<td>0.0</td>
</tr>
<tr>
<td>10</td>
<td>332</td>
<td>342</td>
<td>3.1</td>
</tr>
<tr>
<td>20</td>
<td>361</td>
<td>387</td>
<td>7.2</td>
</tr>
<tr>
<td>30</td>
<td>392</td>
<td>462</td>
<td>17.8</td>
</tr>
</tbody>
</table>
Table 11: Heat flux averages for the top panel surface at time intervals of $t = 0, 10, 20,$ and $30 \text{s}$ for Thornton and ANSYS Fluent laminar solver with convex boundary conditions.

<table>
<thead>
<tr>
<th>Time (s)</th>
<th>Thornton Avg. Heat Flux ($\frac{W}{m^2}$)</th>
<th>Fluent Avg. Heat Flux ($\frac{W}{m^2}$)</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>$4.38 \times 10^4$</td>
<td>$6.09 \times 10^4$</td>
<td>39.0</td>
</tr>
<tr>
<td>10</td>
<td>$4.12 \times 10^4$</td>
<td>$4.11 \times 10^4$</td>
<td>0.2</td>
</tr>
<tr>
<td>20</td>
<td>$4.01 \times 10^4$</td>
<td>$4.23 \times 10^4$</td>
<td>5.5</td>
</tr>
<tr>
<td>30</td>
<td>$3.86 \times 10^4$</td>
<td>$7.50 \times 10^4$</td>
<td>94.2</td>
</tr>
</tbody>
</table>

Table 12: Deflection maximums for the top panel surface at time intervals of $t = 0, 10, 20,$ and $30 \text{s}$ for Thornton and ANSYS Fluent laminar solver with convex boundary conditions.

<table>
<thead>
<tr>
<th>Time (s)</th>
<th>Thornton Deflection (m)</th>
<th>Fluent Deflection (m)</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0.0</td>
</tr>
<tr>
<td>10</td>
<td>$3.30 \times 10^{-4}$</td>
<td>$5.10 \times 10^{-4}$</td>
<td>54.6</td>
</tr>
<tr>
<td>20</td>
<td>$6.10 \times 10^{-4}$</td>
<td>$1.00 \times 10^{-3}$</td>
<td>64.3</td>
</tr>
<tr>
<td>30</td>
<td>$8.38 \times 10^{-4}$</td>
<td>$1.79 \times 10^{-3}$</td>
<td>114.1</td>
</tr>
</tbody>
</table>

These results show that the simulations of this study are predicting somewhat higher values for all measured variables, but producing similar trends for the averaged values. They confirm the conclusion of Thornton and Dechaumphai (1988) that the panel has not yet reached thermal equilibrium up to at least $t = 30 \text{s}$ because temperatures and deflections rise noticeably between each transfer interval.

There are several reasons why the simulations do not match the previous results as well as expected. Differences in meshes is one possibility. Although, efforts were made to include a similar number of layers in the boundary layer, similar amount of elements overall, and the same element type, the meshes were invariably different as exact specifications were not listed by Thornton and Dechaumphai (1988). However, the mesh convergence study discussed in section 2.1 seems to indicate that mesh coarseness should not have had a noticeable effect on the results.

The studies also employed two different numerical methods with Thornton and Dechaumphai (1988) using the finite element method for the fluid solver while ANSYS Fluent
used the finite volume method. As the finite volume method is conservative, it could have given slightly different results than the finite element method which is non-conservative.

Another potential difference could be in the boundary conditions. Thornton and Dechaumphai (1988) used a velocity inlet with fully developed flow whose Reynolds number and initial boundary layer thickness were unknown. The top of the panel was given a sink radiation condition with the sides of the panel conducting heat to the panel supports. However, ANSYS Fluent limitations required radiation to be specified at the bottom of the panel, and a semi-infinite isothermal boundary condition was given to the panel walls in this simulation because the thermal mass of the panel supports was not known. While effort was made to match the original boundary conditions as closely as possible, differences most likely still exist. Simulations with different boundary conditions were being tested at the time of publication to more closely match the results seen in Figure 20. The preliminary results from such simulations can be seen in Figure 33 and Figure 34 in Appendix C.

The thermal analysis was also run slightly differently in this study. Thornton and Dechaumphai (1988) obtained the stabilized flow field at the beginning of transfer interval and then seemed to calculate the heat transfer assuming that the flow field did not change. Due to the initially large thermal gradient between the panel and the viscous flow, the maximum temperature of the fluid would be limited as heat flow to the panel and surrounding air would balance the viscous heat production. Thornton and Dechaumphai (1988) stated that the fluid temperature did not noticeably change throughout the duration of the simulation. In comparison, for this study, the flow field was continuously updated as the thermal analysis proceeded. This allowed the fluid temperature to increase slowly throughout the duration of the simulation in response to a gradual rise in panel temperature.

Lastly, the practice of combining three separate fluid and structural simulations into one loosely coupled simulation could have allowed variance in the results to accumulate with each subsequent transfer interval. Small differences after the initial 10 s could have been magnified with each fluid and structural simulation, causing the last 10 s to be off by quite a large value. This idea is supported by the fact that the temperature and deflection data had worse agreement with each subsequent 10 s simulation. The semi-random assortment of heat flux values heat flux data would seem to contradict this theory, but heat flux at any moment in time is only dependent upon
the instantaneous temperatures in the fluid, and therefore would not be subject to the compounding differences in results.

3.3.2. Reduction of Transfer Interval

In order to understand the validity and accuracy of loosely coupling simulations every 10 s, smaller coupling time steps were chosen. The simulations by Thornton and Dechaumphai (1988) were repeated using addition transfer intervals of 5 s, 2.5 s, and 1 s. They were then compared with each other, and the results from study by Thornton and Dechaumphai (1988) to examine the effect of a decreasing transfer interval. The results for each simulation at \( t = 10 \) s can be seen below in Figure 22, Figure 23, and Table 13.

![Figure 22: Plot of the top panel surface temperature profile vs position for ANSYS Fluent simulations at \( t = 10 \) s for different transfer intervals.](image-url)
Figure 23: Plot of the top panel surface heat flux profile vs position for ANSYS Fluent simulations at $t = 10$ s for different transfer intervals.

Table 13: The deflection, average temperature, and average heat flux at $t = 10$ s for loosely coupled laminar simulations with transfer intervals of 10 s, 5 s, 2.5 s, and 1 s.

<table>
<thead>
<tr>
<th>Transfer Interval (s)</th>
<th>Deflection (m)</th>
<th>Average Temp. (K)</th>
<th>Average Flux ($\frac{W}{m^2}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 (Thornton)</td>
<td>$3.30 \times 10^{-4}$</td>
<td>332</td>
<td>$4.12 \times 10^4$</td>
</tr>
<tr>
<td>10</td>
<td>$5.10 \times 10^{-4}$</td>
<td>342</td>
<td>$4.11 \times 10^4$</td>
</tr>
<tr>
<td>5</td>
<td>$5.37 \times 10^{-4}$</td>
<td>345</td>
<td>$4.73 \times 10^4$</td>
</tr>
<tr>
<td>2.5</td>
<td>$5.68 \times 10^{-4}$</td>
<td>348</td>
<td>$5.50 \times 10^4$</td>
</tr>
<tr>
<td>1</td>
<td>$5.75 \times 10^{-4}$</td>
<td>349</td>
<td>$6.15 \times 10^4$</td>
</tr>
</tbody>
</table>

The results in Table 13 indicate that decreasing the transfer interval predictably affects accuracy. As the magnitude of the results for all three variables increases as the time interval decreases, accounting for panel deflection more frequently seems to increase the overall heat transfer and therefore temperatures through time. These phenomena can also be seen in Figure 22 and Figure 23. Reducing the transfer interval causes the overall temperature and heat transfer at
$t = 10 \, s$ to be higher. The temperature and heat flux profiles are also noticeably higher near the front of the panel for the reduced transfer intervals, giving a nearly flat temperature profile for the 1 s transfer interval scenario.

These higher temperatures and heat fluxes could logically be ascribed to conversion of flow and pressure energy into thermal energy via shock waves and viscous dissipation as these two phenomena would be more prevalent in a panel with more deflection for a given temperature profile. The increased temperature gradients from the shock wave and increased viscous dissipation at the front of the panel with higher deflection would also help to explain the flatter temperature profile for the smaller transfer intervals.

The fact that the magnitude of all three variables consistently increased with a shortening transfer interval has important ramifications for running loosely coupled simulations with hypersonic flow. Most obviously, loosely coupled simulations will never be quite as accurate as fully coupled simulations. However, loosely coupled simulations can of use if the loss of accuracy can be estimated and taken into account. While baseline experimental or fully-coupled simulation data would be needed to exactly define this error, the trends from Table 13 can provide insight on such estimations when such data is not readily available. In a similar manner to a mesh convergence study, a reduction in transfer interval should eventually converge to a maximum transfer interval threshold, below which the solution would not effectively change. The trends in the table do indeed seemed to be trending towards a converged value for the panel deflection and the average temperature. The error for deflection between a transfer interval of 10 s and 5 s is only 5.29 %, and continues to decrease for each subsequent interval. The error follows a similar trend for the average temperature. If another simulation were run with a transfer interval of 0.5 s, the deflection and average temperature at $t = 10 \, s$ would likely be very close to the values for the simulation with a transfer interval of 1 s. Notably, the heat flux does not necessarily seem to be converging. The exact reason for this is not known, but could be the result of an increasingly larger heat flux at the front panel raising the average higher than would otherwise be expected. This phenomenon would arise from an initial shock wave with progressively larger pressure and temperature gradients due to progressively larger deflections. In addition, heat flux only depends upon the instantaneous temperature gradients and therefore would be greater for the smaller transfer interval sizes due to the overall higher temperatures at $t = 10 \, s$
3.3.3. Comparison Between Laminar and Turbulent Models

Loosely coupled simulations were also run using the $k - \epsilon$ model to account for the extra viscous dissipation and convective heat transfer that would exist if the flow were turbulent. The $k - \omega$ model was also used to affirm the accuracy of the $k - \epsilon$ model. Initially, a transfer interval of 10 s to run a full 30 s simulation was going to be used to compare with the results from Thornton and Dechaumphai (1988). However, after running the simulation to 10 s with the $k - \epsilon$ model, the temperatures were found to be far higher than those resulting from the laminar solver. These high temperatures and resulting panel deformations were not only at the limit of validity for the specific heat data and elastic deformation assumption, but also badly warped the mesh elements near the panel surface due to excessive deformation. Therefore, the $k - \epsilon$ model was tested for a transfer interval of 2.5 s. Much higher values were seen for the temperature, heat flux, and deflection than for the laminar model. Comparing the results from the two models in addition to the $k - \omega$ model at $t = 2.5$ s can be seen below in Table 14. Plots of the temperature and heat flux distributions can also be seen in Figure 31 and Figure 32 in Appendix C.

**Table 14:** The deflection, average temperature, and average heat flux at $t = 2.5$ s for laminar, $k - \epsilon$, and $k - \omega$ model simulations.

<table>
<thead>
<tr>
<th>Model</th>
<th>Deflection (m)</th>
<th>Average Temp. (K)</th>
<th>Average Flux ($\frac{W}{m^2}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Laminar</td>
<td>$2.978 \times 10^{-4}$</td>
<td>311</td>
<td>$5.34 \times 10^4$</td>
</tr>
<tr>
<td>$k - \epsilon$</td>
<td>$3.017 \times 10^{-3}$</td>
<td>596</td>
<td>$8.63 \times 10^5$</td>
</tr>
<tr>
<td>$k - \omega$</td>
<td>$2.933 \times 10^{-3}$</td>
<td>586</td>
<td>$8.22 \times 10^5$</td>
</tr>
</tbody>
</table>

The $k - \epsilon$ and $k - \omega$ models agree fairly well, with the $k - \omega$ model yielding slightly more conservative results. However, even the largest discrepancy found between the values of average flux was only 4.75 %. If the entirety of the flow is indeed turbulent, then confidence is felt predicting its characteristics using the $k - \epsilon$ model.

Both turbulence models have values well above those of the laminar model. The large difference in values between the laminar and two turbulence models is most likely due to the added effects of turbulence as discussed previously. While such a discrepancy is concerning, it is not surprising given the different formulations used to define viscous dissipation and heat transfer in
the laminar and turbulence models. It also highlights the need for a solver to accurately calculate the entire range of physics for a hypersonic flow problem, as the high energy in the flow can manifest itself in different forms. Choosing the incorrect model could result in serious errors.

As discussed in section previously, Thornton and Dechaumphai (1988) did not include an initial Reynolds number nor the length the flow travelled prior to reaching the inlet, and a critical Reynolds number could not be defined for this scenario. Therefore, it cannot be definitively determined whether the flow field they examined was turbulent. This suggests that further investigation is needed to determine if the usage of a laminar solver was valid for the given scenario. If the results from the $k - \varepsilon$ and $k - \omega$ models are indeed more realistic than those from the laminar model, then the temperatures and deformation behavior of the panel were drastically under predicted.

### 3.3.4. Concave Simulations

Thornton and Dechaumphai (1988) also ran similar simulations for a panel with pins on the top corners leading to concave deformation. While they provided no data to use for comparison, an image of the density field was provided for the results of the loosely coupled simulation up to $t = 30$ s. For this study, a similar loosely coupled simulation for the concave panel setup was run for 30 s using the same process as outlined for the convex panel setup. This simulation was done for completeness and to ensure that the concave panel setup behaved in a similar manner to the convex panel setup. The resulting density fields for the convex and concave panel setups from Thornton and Dechaumphai (1988) and from the simulations can be seen below in Figure 24.
Figure 24: Plots of the density fields at $t = 30\ s$ for (A) Thornton and Dechaumphai (1988) and (B) ANSYS Fluent simulations for convex (top) and concave (bottom) panel setups.

Although the units shown in the density field used for this study do not match those used by Thornton and Dechaumphai (1988), the shock wave and expansion wave behavior match well. Higher densities can be clearly seen at compression corners indicating shock waves. A similar density decrease can be seen at expansion corners showing expansion waves. Unfortunately, more data is not available, but the similarity in the overall density trends including shock angles between this study and the one by Thornton and Dechaumphai (1988) for both the concave and convex panel setups is encouraging. Further investigation of concave simulations is left for future work.

3.3.5. Buckling Simulations

The panel in all of the simulations previously discussed underwent smooth deformations with no discontinuities other than those introduced by only updating the deformation at discrete time intervals. Unfortunately, panel deformation in actual hypersonic flow is often not a smooth process. The boundary conditions and thermal loadings of panels such as those found on hypersonic aircraft can often lead to sudden buckling behavior. Predictions for buckling loads and temperatures were discussed in section 3.2. Critical temperature changes that would cause buckling were found to be $\Delta T_{crit} = 47.0\ K$ and $\Delta T_{crit} = 166.5\ K$ for pinned and fixed end.
boundary conditions respectively. Loosely coupled simulations indicated that temperature rises of $\Delta T = 462 - 295 = 167 K$ after 30 s would be high enough to induce buckling in the pinned end panel, and would possibly induce buckling in the fixed end panel. As buckling in the fixed end panel would only be found at the very end of the 30 s considered for this study, only buckling for the pinned end panel was considered. However, the process would be nearly identical for both types of buckling.

In order to simulate pinned-end buckling using loosely coupled simulations, a UDF had to be implemented. For this study, the time step had to be drastically reduced every time a UDF deformation was implemented to allow the flow time to stabilize as described in section 2.5. As the current UDF is not capable of updating the time step, the simulation was run until the average temperature change was $\Delta T_{\text{crit}} = 47.0 K$ at an average temperature seen below in Equation (49).

$$T_{\text{buckle}} = T_{\text{initial}} + \Delta T_{\text{crit}} = 295.0 K + 47.0 K = 342.0 K \quad (49)$$

This coincidentally happened to be the average temperature on the panel with a laminar solver at $t = 10 s$ as seen in Table 10. The simulation was therefore run for 10 s, at which point the UDF was introduced to deform the panel according to the results of the buckling analysis. After the buckling had occurred, the flow was allowed to stabilize. As with the $k - \epsilon$ model run for 10 s, the resulting temperatures were so high that the temperature dependent data for specific heat was no longer valid, preventing any thermal analysis following the buckling.

Comparing this buckling simulation with the coupled simulation displays the importance of understanding the panel’s boundary conditions, especially when running loosely coupled simulations. The temperature profiles from the two simulations immediately following deformations at 10 s can be seen below in Figure 25.
Figure 25: Plot of the top panel surface temperature vs horizontal position for ANSYS Fluent laminar simulations for the pinned support panel and convex panel setup at $t = 10$ s following the deformation update by UDF.

The temperature profile from the pinned support panel that just buckled is much higher than that of the panel with the convex deformation setup. While the panel in both cases deformed upward into the flowing causing shock waves, the buckling panel deformed much further causing a large temperature spike at the beginning of the panel which combined with heightened viscous dissipation to yield higher temperatures overall. The peak temperature at 943 $K$ is in fact so close to the maximum valid temperature of 1,000 $K$ for the specific heat values that further simulation could only be attempted with extended specific heat data. Additionally, care would have to be taken to ensure that disassociation would not occur and negate the continuum assumption. Overall, care must be taken to determine buckling behavior accurately so that the temperature and other properties of the flow can be successfully predicted.
Chapter 4: Conclusion

Loosely coupled simulations offer a relatively computationally inexpensive technique for examining fluid-thermal-structural interactions in hypersonic flow over panels. However, attention must be given to the time interval at which data is transferred between systems. Accuracy can become a concern if the transfer interval is too large.

Thornton and Dechaumphai (1988) undertook such an approach a few decades ago in examining spacecraft panel behavior during flight. Their approach was innovative and successfully implemented the loosely coupled technique with a laminar solver. Due to the time period of the study however, the team was no doubt limited in the amount of computing power available and so were limited to a relatively large data transfer interval.

This study attempted to replicate their results by using ANSYS Fluent and Abaqus FEA to run fluid-thermal and structural simulations. Steady state fluid simulations were first run to compare turbulence solvers and fluid property formulations. Comparing turbulent and non-turbulent models was especially important as the flow in the domain study was possibly turbulent, but was initially studied using a laminar solver. Geometry with discrete angles and curved angles was examined and compared to oblique shock theory using a MATLAB model. These steady state results led to the use of laminar and $k - \varepsilon$ models with temperature dependent fluid property formulations similar to those used by Thornton and Dechaumphai (1988).

Structural simulations were examined next to determine deflections and buckling caused by thermal loading in the panel. Eccentric loading due to thermal gradients through the thickness of the panel was considered, but was found to have a negligible effect on critical buckling temperatures. Euler buckling theory was therefore used as comparison for the structural results.

Finally, loosely coupled simulations were completed. A fluid-thermal simulation was run for 10 s to determine temperature profiles in the panel. The panel was analyzed structurally to determine the vertical deflection caused by the profiles. A UDF was then implemented to update the panel deformation in the fluid-thermal simulation. The process was done three times for a full 30 s simulation. This process was repeated several times for different scenarios consisting of the laminar and $k - \varepsilon$ solvers, and smaller transfer intervals to see their effect on the accuracy of the results.

The results of the laminar solver were compared to the results from Thornton and Dechaumphai (1988). While deflections, temperatures, and heat fluxes were consistently over
predicted in this study, the results of the two studies shared similar trends through time. The differences between the two studies are attributed to differing numerical methods, boundary conditions, initial conditions, and material properties. Additionally, the simulations from this study continuously updated the thermal characteristics of the flow while the simulations by Thornton and Dechaumphai (1988) did not. This could have allowed variances in the results to compound over subsequent simulations.

Differences between the laminar model and turbulence models were also investigated. The \( k - \epsilon \) and \( k - \omega \) models predicted temperatures about 300 – 400 K higher than those predicted by the laminar solver in this study and in the study by Thornton and Dechaumphai (1988). Such large differences are concerning and indicate that care must be taken when analyzing a fluid dynamics problem and selecting the correct solver. Further investigation is needed to accurately determine the critical Reynolds number and transition length for this flow. Combining this lack of knowledge about the flow transition with uncertainty of initial conditions makes it difficult to draw definite conclusions about the results of Thornton and Dechaumphai (1988).

The transfer interval study was more fruitful. It was found that decreasing the time transfer interval had a large impact on the overall results of the simulation. This was not surprising as panel deformation and buckling during hypersonic flight make up a system whose fluid, thermal, and structural components are highly coupled. Four different transfer intervals were considered and used to predict flow properties at \( t = 10 \) s. Overall, the smaller intervals lead to larger deflections and higher temperatures due to the panel deflection being updated more frequently. However, as the transfer intervals decreased, so did the error between their results. This indicated that a convergence study could be done to find a transfer interval that more closely matches the actual results without requiring large amounts of computational time. While experimental or fully coupled simulation results would be required to optimize the transfer interval fully, a convergence study using just loosely coupled simulations shows promise to find a nominally efficient solution.

Loosely coupled simulations for a panel with the concave and pinned end setups were also run. While comparisons were not made with these simulations due to lack of data, they qualitatively matched the results from Thornton and Dechaumphai (1988) and the convex panel setup simulations. It was found that buckling and concave boundary conditions could also successfully be analyzed with loosely coupled simulations.
Ultimately, while decreasing the data transfer interval can help increase the accuracy of hypersonic simulations, the computational and physical time needed for the simulations increases quickly. For this study, transferring data every few seconds in the simulation was time intensive and unwieldy. A more streamlined coupling method would have certainly improved the situation, but large time investments remain. Reduced order modeling through POD can offer an even faster alternative to loosely coupled simulations. Once data is generated for one case, a POD ROM could be constructed and used for iterative design, allowing parametric studies of dimensions, velocities, and material properties to be conducted. Further study is needed to judge the accuracy of ROMs constructed from the data from this study’s results, but the data were generated successfully as the simulations progressed in time. Future work involves creating POD ROMs from data from this study using the code created by Caraballo et al.
Chapter 5: Future Work

The work done for this study has involved analyzing loosely coupled hypersonic simulations. The results show that the method can be used to analyze hypersonic flow if extreme accuracy is not needed. This method is ideal for iterative design in the initial stages of a hypersonic study or aircraft design. However, future work would involve fully coupling the fluid and structural solver to ensure that the physical coupling of the two systems is fully taken into account. Direct validation against an experimental data set would also be useful.

The results from the loosely coupled simulations can now be used to provide data for a POD ROM. The accuracy of such a ROM would then need to be studied. Potential work to build upon this study can be seen below:

- Test data from uncoupled and loosely coupled simulations on the POD code by Caraballo et al.
- Investigate different boundary conditions such as walls with constant heat flux to more closely match the heat flux results from Thornton and Dechaumphai (1988).
- Investigate smaller transfer intervals to see simulation results continue to converge.
- Investigate the effects of transfer interval size on loosely coupled simulations for a panel with concave and buckling setups.
- Increase the operating range of the simulations by increasing the range of the specific heat values.
- Redefine the mesh correction scheme to allow larger deformations to be examined without compromising accuracy.
- Run fully coupled simulations to account for the physical coupling in a hypersonic system.
- Validate against experiment data set.
Chapter 6: References


Appendix

Appendix A: MATLAB Program Procedure Outline

Figure 26: MATLAB program procedure flow diagram.

Appendix B: Mesh Images

Mesh A (used for steady state results) and mesh B (used for loosely coupled results) are shown below in Figure 27 (A) and Figure 27 (B) respectively.
Figure 27: Images of (A) mesh A and (B) mesh B.

Appendix C: Simulation Plots

Plots from simulations run in this study can be seen below in the following figures.
Figure 28: Plot of the top panel surface temperature profile vs position at time increments of $t = 10, 20, \text{ and } 30 \text{ s}$ for convex boundary conditions (SI units).

Figure 29: Plot of the top panel surface heat flux profile vs position at time increments of $t = 10, 20, \text{ and } 30 \text{ s}$ for convex boundary conditions (SI units).
Figure 30: Plot of static pressure at times of $t = 10\, s$ (A), $20\, s$ (B), and $30\, s$ (C) for the ANSYS Fluent laminar solver with convex boundary conditions.
**Figure 31:** Plot of the top panel surface heat flux profile vs position for the ANSYS Fluent laminar, $k - \epsilon$, and $k - \omega$ models for convex boundary conditions at $t = 2.5$ s.

**Figure 32:** Plot of the top panel surface temperature profile vs position for the ANSYS Fluent laminar, $k - \epsilon$, and $k - \omega$ models for convex boundary conditions at $t = 2.5$ s.
Simulations being run at the time of publishing attempted to more closely match the results of Thornton and Dechaumphai (1988) by changing the boundary conditions of the ANSYS Fluent simulations. Figure 33 below shows the results from replacing the isothermal boundary condition of the walls preceding and following the panel with a constant heat flux of $q = 43,720 \text{ W/m}^2$. The heat flux distribution more closely matches the results seen in Figure 20, but the temperature distribution is even less accurate.

![Graph showing temperature and heat flux profiles](image)

**Figure 33**: Plot of (A) the top panel surface temperature profile vs position and of (B) the top panel surface heat flux profile vs position for ANSYS Fluent laminar simulation at $t = 10 \text{ s}$ with the walls preceding and following the panel having constant heat flux $q = 43,720 \text{ W/m}^2$.

Figure 34 shows the results of running a simulation with a predefined inlet temperature profile based on Equation (67). It can be seen that such a change mostly preserves the temperature profile while yielding a heat flux profile similar to the one in Figure 20. The spikes in heat flux at the ends of the panel remain, but are likely artificial spikes caused by the isothermal conditions on the ends of the panel.
Figure 34: Plot of (A) the top panel surface temperature profile vs position and of (B) the top panel surface heat flux profile vs position for ANSYS Fluent laminar simulation at \( t = 10 \, s \) with the inlet having a predefined temperature profile.

Appendix D: Derivations

D.1 Buckling Equations

Consider Hooke’s law for stress and strain, and the equation relating strain to the coefficient of thermal expansion

\[
\sigma = E\epsilon \quad (50)
\]

\[
\epsilon = \alpha \Delta T \quad (51)
\]

Assuming a constant area, and combining and rearranging these two equations yields Equation (44)

\[
\Delta T_{\text{comp}} = \frac{\sigma}{E\alpha} = \frac{F}{AE\alpha} = \frac{F_{\text{buckle}}}{wt_{\text{pan}}E\alpha}
\]

The critical buckling force for Euler buckling is given in Equation (38) as
\[ F_{buckle} = \frac{C\pi^2EI}{L^2} \]

Combining Equation (44) and Equation (38) yields

\[ \Delta T = \left( \frac{C\pi^2EI}{L^2} \right) \frac{w t_{pan}}{wt_{pan}E\alpha} \quad (52) \]

Since the panel has a constant rectangular area, the second moment of inertia is

\[ I = \frac{1}{12} t_{pan}^3 w \quad (53) \]

Plugging Equation (53) into Equation (52) and rearranging yields Equation (42)

\[ \Delta T_{Euler} = \frac{C\pi^2 t_{pan}^2}{12\alpha L^2} \]

Similarly, combining Equation (44) and Equation (41) gives Equation (43) below

\[ \Delta T_{secant} = \frac{\sigma_{yield}}{E\alpha \left( 1 + \left( \frac{e_{load}}{2k_{gyr}} \right) \sec \left( \left( \frac{L}{2k_{gyr}} \right) \sqrt{\Delta T\alpha} \right) \right)} \]

**D.2 Panel Thermal Eccentric Loading**

Consider the panel in Figure 35 having top surface temperature \( T_{top} \) and bottom surface temperature \( T_{bot} \), and assume the temperature varies linearly through the thickness of the panel.
This implies

\[ T(y) = Ay + B \]

\[ T\left(\frac{t_{pan}}{2}\right) = T_{top} = A\left(\frac{t_{pan}}{2}\right) + B \]

\[ T\left(-\frac{t_{pan}}{2}\right) = T_{bot} = A\left(-\frac{t_{pan}}{2}\right) + B \]

Solving for \(A\) and \(B\) yields

\[ T(y) = \left(\frac{T_{top} - T_{bot}}{t_{pan}}\right)y + \left(\frac{T_{top} + T_{bot}}{2}\right) \quad (54) \]

Combining this expression with Equation (50) and Equation (51) gives

\[ \sigma(y) = \Delta T(y) \alpha E = (T(y) - T_0) \alpha E \quad (55) \]
where $T_0 = 295 \, K$ is the initial panel temperature and $\sigma(y)$ is the stress in the panel for a given $y$. Integrating Equation (55) and assuming isotropic material constants gives the total force within the panel

$$ F = \int \int \sigma(y) \, dy \, dz = \int \int (T(y) - T_0) \alpha E \, dy \, dz = w \alpha E \int (T(y) - T_0) \, dy $$

Combining Equation (54) and Equation (56) and integrating yields

$$ F = w \alpha E \left[ \left( \frac{T_{top} - T_{bot}}{2t_{pan}} \right) y^2 + \left( \frac{T_{top} + T_{bot}}{2} - T_0 \right) y \right]_{-\frac{t_{pan}}{2}}^{\frac{t_{pan}}{2}} $$

The total force in Equation (62) through the entire panel thickness is centered at some distance $e_{load}$ from the x-axis which is defined as the loading eccentricity. This force can be divided into two equal forces above and below $e_{load}$ defined respectively as $F_{top}$ and $F_{bot}$ and written as

$$ F_{top} = w \alpha E \left[ \left( \frac{T_{top} - T_{bot}}{2t_{pan}} \right) y^2 + \left( \frac{T_{top} + T_{bot}}{2} - T_0 \right) y \right]_{0}^{\frac{t_{pan}}{2}} $$

$$ F_{bot} = w \alpha E \left[ \left( \frac{T_{top} - T_{bot}}{2t_{pan}} \right) y^2 + \left( \frac{T_{top} + T_{bot}}{2} - T_0 \right) y \right]_{-\frac{t_{pan}}{2}}^{0} $$

Solving these equations, setting them equal to each other, and cancelling common terms yields
This can further be written in quadratic form

\[ e_{load}^2 \left( \frac{T_{top} - T_{bot}}{2t_{pan}} \right) + \frac{2}{4} t_{pan}^2 + \frac{2}{2} (T_{top} + \frac{T_{bot}}{2} - T_0) \left( -2e_{load} \right) = 0 \]  

(60)

which combined with the quadratic equation gives the eccentricity in Equation (47) as

\[ e_{load} = \frac{\left( T_0 - \frac{T_{top} + T_{bot}}{2} \right) + \sqrt{\left( \frac{T_{top} + T_{bot}}{2} - T_0 \right)^2 - \left( \frac{T_{top} - T_{bot}}{2} \right)^2}}{\left( \frac{T_{top} - T_{bot}}{t_{pan}} \right)} \]  

(61)

D.3 Trapezoidal Corner Angle Equation

Consider a convex deformed panel in the shape of an arc with length \( L \) and vertical deflection \( d \). The panel arc is assumed to be a scaled half circle with a radius of \( L/2 \). If \( x \) is the horizontal dimension and \( y \) is the vertical dimension with the origin at the center of the top surface of the undeformed panel, the general equation for the shape of the arc is given as

\[ y = \frac{d}{L/2} \sqrt{\left( \frac{L}{2} \right)^2 - x^2} \]  

(62)

where \( \frac{d}{L/2} \) is the scaling factor to ensure that the arc is the correct height.

Setting this equation equal to \( d/2 \) gives the \( x \) distance from the origin where the arc \( y \) value is exactly half of the deflection yielding
\[ x_{1/2} = \sqrt{\left(\frac{L}{2}\right)^2 - \left(\frac{d}{2} \cdot \frac{L/2}{d}\right)^2} = \sqrt{\frac{L^2}{4} - \frac{L^2}{16}} = \pm \frac{L \sqrt{3}}{4} \quad (63) \]

The deflection angle \( \theta \) can then be found trigonometrically using a right triangle with half the deflection on one leg and \( L/2 \) minus \( x_{1/2} \) on the other leg. Solving for \( \theta \) gives

\[ \theta = \tan^{-1} \left( \frac{d/2}{\frac{L}{2} - x_{1/2}} \right) = \tan^{-1} \left( \frac{d/2}{\frac{L}{4} (2 - \sqrt{3})} \right) = \tan^{-1} \left( \frac{2d}{L (2 - \sqrt{3})} \right) \quad (64) \]

D.4 Sinusoidally Driven Oscillations

The UDF for sinusoidal oscillations was designed to take the angular velocity, the maximum panel deflection, and the panel length as inputs used to define the vertical deflection \( y \). Consider Equation (62) which gives the scaled half circle deflection of the panel. Since sine is a function that varies only between \(-1\) and \(1\), multiplying by \( \sin(\omega t) \) will periodically deform a function between its maximum value and the negative of its maximum value. Using this fact and plugging in \( d = d_{\text{max}} \) yields

\[ y = \frac{2d_{\text{max}}}{L_{\text{pan}}} \sqrt{\left(\frac{L_{\text{pan}}}{2}\right)^2 - x^2} \cdot \sin(\omega t) \quad (65) \]

where \( d_{\text{max}} \) is the maximum deflection, \( L_{\text{pan}} \) is the panel length, \( x \) is the horizontal position along the panel, \( \omega \) is some angular velocity, and \( t \) is the time. Note that \( y \) is only realistically defined in the range \[ \frac{L_{\text{pan}}}{2} \leq x \leq \frac{L_{\text{pan}}}{2} \].

D.5 Inlet Boundary Layer Velocity Equation

For the loosely coupled simulations, the inlet was given a velocity profile of fully developed flow. A preliminary laminar, steady state simulation was run on mesh A at 1443 \( m/s \) to let the boundary layer fully develop. The velocity profile was then sampled along a vertical line
positioned 0.1397 m (5.5 in) from the entrance and a cubic equation was fit to the data within the boundary layer given as

\[ v_{\text{boundary}} = (-2.02 \times 10^{13})y^3 + (-1.08 \times 10^{10})y^2 + (9.65 \times 10^6)y \quad (66) \]

This equation was then implemented into a UDF to use in ANSYS Fluent for the boundary layer with the remaining portion of the inlet given a constant velocity of 1443 m/s. Since this velocity profile was sampled 0.1397 m from freestream flow, the Reynolds number per length calculated in Equation (4) gives a Reynolds number of

\[ Re = \frac{Re}{L} * L = (9.493 \times 10^7 \text{ m}^{-1})(0.1397 \text{ m}) = 1.326 \times 10^7 \]

which potentially makes the flow fully turbulent or transitioning at the inlet.

**D.6 Inlet Boundary Layer Temperature Equation**

Most of the loosely coupled simulations in this study were run using a constant temperature of \( T = 295 \text{ K} \) at the inlet with the velocity profile described above. However, simulations were being run at the time of publication with attempts to more closely match the results of Thornton and Dechaumphai (1988). One such attempt involved adding a temperature profile at the inlet of the domain. As in section D.5 a preliminary laminar, steady state simulation was run on mesh A at 1443 m/s to let the boundary layer fully develop. The temperature was then sampled along a vertical line positioned 0.1397 m (5.5 in) from the entrance of the domain and the boundary layer data was fit to an empirical quartic equation of the form

\[ T_{\text{boundary}} = (-7.87 \times 10^{16})y^4 + (7.22 \times 10^{13})y^3 + (-2.24 \times 10^{10})y^2 + (2.34 \times 10^6)y + 296 \quad (67) \]

This equation was then implemented into a UDF to use in ANSYS Fluent for the boundary layer with the remaining portion of the inlet given a constant temperature of 295 K.
Appendix E: Proper Orthogonal Decomposition

POD uses inner products to form ROMs as discussed in section 1.6. Further explanation of using the theory to model turbulence is given as a reference.

Of particular interest in turbulence modeling are fluctuations in the mean flow as they are the hardest phenomena for CFD to capture. Non-dimensional velocity fluctuations are represented as \( \hat{\mathbf{u}}'(\mathbf{x}, t) = [\hat{u}', \hat{v}', \hat{c}'] \) and are approximated below as

\[
\hat{\mathbf{u}}'(\mathbf{x}, t) \approx \sum_{i=1}^{N_{\#}} a_i(t) \varphi_i(\mathbf{x})
\]

where \( \varphi_i(\mathbf{x}) \) is the POD modes, \( a_i(t) \) is the amplitudes of the POD modes, and \( N_{\#} < M_{\#} \) is the number of modes captured using \( M_{\#} \) snapshots of the flow. A snapshot is defined as the flow field at a discrete point in time. The POD modes are further defined by

\[
\varphi_i(\mathbf{x}) = \sum_{k=1}^{M_{\#}} A_{ik}(t) \hat{\mathbf{u}}'(\mathbf{x}, t_k)
\]

where \( \hat{\mathbf{u}}'(\mathbf{x}, t_k) \) is the velocity fluctuation data at discrete snapshots \( k \), and \( A_{ik}(t) \) is the solution to the eigenvalue problem

\[
C(t, t_k)A = \lambda A
\]

with \( C(t, t_k) \) being a tensor that correlates velocity profiles in snapshots \( k \) to the profiles in the snapshot at time \( t \) by

\[
C(t, t_k) = \frac{1}{M_{\#}} \langle \hat{\mathbf{u}}'(\mathbf{x}, t), \hat{\mathbf{u}}'(\mathbf{x}, t_k) \rangle_s
\]
Note that \( C(t, t_k) \) is integrated spatially over \( S \) and is not dependent upon the number of grid points sampled. The only remaining unknown now is \( a_i(t) \) from Equation (68), but it can be defined as

\[
a_i(t) = \int_S \hat{u}'(\vec{x}, t) \varphi_i(\vec{x}) d\vec{x}
\]  

(72)

where \( \hat{u}'(\vec{x}, t) \) is taken from a known flow field and \( \varphi_i(\vec{x}) \) was defined above in Equation (69). Note that the number of POD modes \( N \) is only limited by the number of snapshots \( M \) and not by the number of grid points sampled in the domain. More snapshots allow more modes to be captured and increases the accuracy of the decomposition. Further detail on the POD process is out of the scope of this study and can be found in the study by Caraballo et al. (2007).

**Appendix F: Comparison of Thermal Conductivity Formulations**

The kinetic theory and Prandtl number formulations for calculating thermal conductivity were compared to examine the differences in values generated by the two methods. The results can be seen below in Figure 36. A constant value of \( Pr = 0.72 \) was used in the Prandtl number formulation to calculate the thermal conductivity and was compared to temperature dependent Prandtl number values. The error caused by assuming this constant value was also calculated. These comparisons can be seen below in Figure 37.
Figure 36: Plot of thermal conductivity vs temperature for Prandtl number and kinetic theory formulations.

Figure 37: Plot of (A) empirical Prandtl number data vs temperature and of (B) error between $Pr = 0.72$ assumption and empirical Prandtl number data vs temperature.
Appendix G:  Validity of Elastic Deformation Assumption

Metals can be approximated as having a linear stress versus strain relationship by Hooke’s law in the elastic region of strain. Such an approximation is valid as long as the von Mises stresses in the material stay below the yield stress. As the assumption of linear deformation was used in this study when running quasi-static structural simulations, the panel needed to be examined to make sure it stays within the elastic region. Therefore, the von Mises stresses from the top and bottom of the panel were extracted when they were the highest at \( t = 30 \, s \) with the laminar model and plotted along with a line representing the yield stress. It can be seen in Figure 38 that the von Mises stresses are indeed below the yield stress along the length of the panel.

The one exception lies at the ends of the panel on the bottom where the pins are located. However, this exceptionally high stress is an artificial phenomenon caused by attempting to represent an infinitely small pinned point with a finite mesh. Further mesh refinement in the panel would reduce these extreme stresses, but would not noticeably change the results along the rest of the panel. Thornton and Dechaumphai (1988) note a similar effect in their stress results, and similarly attribute it to the coarseness of the mesh.
Figure 38: Plot of von Mises stress vs horizontal position for the top and bottom surface of panel for loosely coupled simulation at $t = 30 \, s$ for the ANSYS Fluent laminar solver.

Appendix H: Material Properties

Table 15: Constant properties of AM-350 stainless steel from High Temp Metals (2015).

<table>
<thead>
<tr>
<th>Density</th>
<th>Specific Heat</th>
<th>Yield Stress</th>
<th>Poisson's Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\frac{kg}{m^3}$</td>
<td>$\frac{J}{kg \cdot K}$</td>
<td>$Pa$</td>
<td>$None$</td>
</tr>
<tr>
<td>7,810</td>
<td>502.416</td>
<td>$9.79 \times 10^8$</td>
<td>0.3</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Temperature (K)</th>
<th>Thermal Conductivity $\frac{W}{m^2 \cdot K}$</th>
<th>Thermal Expansion $K^{-1}$</th>
<th>Modulus of Elasticity $Pa$</th>
</tr>
</thead>
<tbody>
<tr>
<td>311</td>
<td>14.5</td>
<td>1.13 $\times 10^{-5}$</td>
<td>300</td>
</tr>
<tr>
<td>366</td>
<td>15.4</td>
<td>1.22 $\times 10^{-5}$</td>
<td>477</td>
</tr>
<tr>
<td>422</td>
<td>16.2</td>
<td>1.26 $\times 10^{-5}$</td>
<td>589</td>
</tr>
<tr>
<td>477</td>
<td>17.0</td>
<td>1.30 $\times 10^{-5}$</td>
<td>644</td>
</tr>
<tr>
<td>533</td>
<td>17.8</td>
<td>1.30 $\times 10^{-5}$</td>
<td>700</td>
</tr>
<tr>
<td>589</td>
<td>18.7</td>
<td>1.21 $\times 10^{-5}$</td>
<td></td>
</tr>
<tr>
<td>644</td>
<td>19.6</td>
<td>1.26 $\times 10^{-5}$</td>
<td></td>
</tr>
<tr>
<td>700</td>
<td>20.3</td>
<td>1.35 $\times 10^{-5}$</td>
<td></td>
</tr>
<tr>
<td>755</td>
<td>21.1</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 17: Temperature dependent specific heat properties of air (Cengel & Boles, 2002).

<table>
<thead>
<tr>
<th>Temperature (K)</th>
<th>$C_p$ $\left(\frac{J}{kg \cdot K}\right)$</th>
<th>$C_v$ $\left(\frac{J}{kg \cdot K}\right)$</th>
<th>$k = \frac{C_p}{C_v}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>250</td>
<td>1,003</td>
<td>716</td>
<td>1.401</td>
</tr>
<tr>
<td>300</td>
<td>1,005</td>
<td>718</td>
<td>1.400</td>
</tr>
<tr>
<td>350</td>
<td>1,008</td>
<td>721</td>
<td>1.398</td>
</tr>
<tr>
<td>400</td>
<td>1,013</td>
<td>726</td>
<td>1.395</td>
</tr>
<tr>
<td>450</td>
<td>1,020</td>
<td>733</td>
<td>1.392</td>
</tr>
<tr>
<td>500</td>
<td>1,029</td>
<td>742</td>
<td>1.387</td>
</tr>
<tr>
<td>550</td>
<td>1,040</td>
<td>753</td>
<td>1.381</td>
</tr>
<tr>
<td>600</td>
<td>1,051</td>
<td>764</td>
<td>1.376</td>
</tr>
<tr>
<td>650</td>
<td>1,063</td>
<td>776</td>
<td>1.370</td>
</tr>
<tr>
<td>700</td>
<td>1,075</td>
<td>788</td>
<td>1.364</td>
</tr>
<tr>
<td>750</td>
<td>1,087</td>
<td>800</td>
<td>1.359</td>
</tr>
<tr>
<td>800</td>
<td>1,099</td>
<td>812</td>
<td>1.353</td>
</tr>
<tr>
<td>900</td>
<td>1,121</td>
<td>834</td>
<td>1.344</td>
</tr>
<tr>
<td>1,000</td>
<td>1,142</td>
<td>855</td>
<td>1.336</td>
</tr>
</tbody>
</table>