SIZING WIND TUNNEL HEATER FOR HIGH ENTHALPY CONDITIONS

A Thesis

presented to

the Faculty of California Polytechnic State University,

San Luis Obispo

In Partial Fulfillment

of the Requirements for the Degree

Master of Science in Mechanical Engineering

by

Justin Slavick

June 2024
COMMITTEE MEMBERSHIP

TITLE:  Sizing Wind Tunnel Heater for High Enthalpy Conditions

AUTHOR:  Justin Slavick

DATE SUBMITTED:  June 2024

COMMITTEE CHAIR:  Kim Shollenberger, Ph.D.
Professor of Mechanical Engineering

COMMITTEE MEMBER:  Russell Westphal, Ph.D.
Professor of Mechanical Engineering

COMMITTEE MEMBER:  Nandeesh Hiremath, Ph.D.
Assistant Professor of Aerospace Engineering
This paper determines the feasibility of adding a heater to an existing blowdown supersonic wind tunnel to unlock new high-enthalpy test applications, considering cost and power requirements at a variety of different states. This process includes both modeling the current range of test section properties in Cal Poly’s blowdown wind tunnel and determining the new range of properties that a heat exchanger could induce. These results are verified with a computational fluid dynamics study. Additionally, sublimation and ablation properties of materials are explored to create appropriate models to study atmospheric re-entry once the heat exchanger is implemented.

It is found that adding a heater to the supersonic wind tunnel would significantly increase the test section temperature. Additionally, enough heat could be added without damaging the facility to surpass the vapor pressure of camphor and naphthalene at test section conditions, allowing for the tunnel to be used for sublimation and ablation applications. Using the tunnel with the variable Mach nozzle currently installed would induce minimum heater power requirements of 75kW for a Mach 4 configuration and 200kW for the testing Mach 3.13 condition to reach this vapor pressure. However, this power requirement can be significantly reduced by installing a new nozzle that would induce flow at a Mach number of 6-8. Liquefaction is found to be avoided at every test and Mach condition, even without any heat added, while condensation cannot be avoided at any configuration, regardless of nozzle used or heat added. Therefore, we recommend that a dryer be installed to help remedy these issues.
I would firstly like to thank Dr. Kim Shollenberger for her willingness to assume the duties of Chair and guiding me throughout this thesis. You not only have provided me with extensive support in my research and analysis but have also remained extremely patient in helping me with the logistical portions of this project.

I would also like to thank my advisor, Dr. Nandeesh Hiremath, for mentoring me as I underwent this project. Not only have you also been so incredibly patient and supportive, but you have taught me a lot of useful skills in research, technical paper writing, and aerospace engineering that will remain helpful far into the future.

I would like to thank Dr. Russ Westphal for his willingness to act as my third committee member on such a short time frame and for the valuable insight he provided towards this project.

I would also like to thank Dr. Dianne DeTurris for providing me with all documentation for the supersonic wind tunnel and supporting us in our efforts to upgrade the facility to support hypersonic testing. I thank fellow student Ben Kenworthy for introducing me to the supersonic wind tunnel and helping us understand how it operates. Finally, I thank the shop technician, Cody Thompson, for helping us run the tunnel, maintain its operation, and strategize the purchase and installation of new components.

Last but not least, I would like to thank my family for their thorough and continuous encouragement of me as I tackled this thesis. They can never fail to keep me driving forward and always provide a backbone to me achieving my dreams and aspirations.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>LIST OF TABLES</th>
<th>viii</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIST OF FIGURES</td>
<td>ix</td>
</tr>
</tbody>
</table>

## CHAPTER

1. **Introduction** ......................................................... 1
   - 1.1 Existing Hypersonic Ground Testing Facilities and Applications ... 2
     - 1.1.1 Low-Enthalpy Applications .................................. 3
     - 1.1.2 Low-Enthalpy Facilities .................................... 6
     - 1.1.3 High-Enthalpy Applications ................................. 7
     - 1.1.4 High-Enthalpy Facilities .................................. 9
     - 1.1.5 The Need for Additional Facilities ....................... 11
   - 1.2 The Supersonic Wind Tunnel at Cal Poly ....................... 12

2. **Heater Sizing Constraints** ........................................... 16
   - 2.1 Selecting a Temperature Range .................................. 16
   - 2.2 Modeling Sublimation ............................................ 18
   - 2.3 Thermodynamic Model ............................................ 19

3. **Analytical Results** .................................................. 24
   - 3.1 Current Nozzle .................................................. 24
   - 3.2 Decreased Throat Area .......................................... 28

4. **Computational Fluid Dynamics Study** ............................... 33
   - 4.1 Assumptions ..................................................... 33
   - 4.2 Constructing the Model ......................................... 36
   - 4.3 Running the Simulation ......................................... 37
<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1</td>
<td>Pressure, static temperature, and Mach number comparison between the analytical and CFD results for the non-heated case.</td>
<td>41</td>
</tr>
<tr>
<td>4.2</td>
<td>Pressure, static temperature, and Mach number comparison between the analytical and CFD results for the heated case.</td>
<td>44</td>
</tr>
<tr>
<td>5.1</td>
<td>Test section temperature, maximum model surface temperature, and heating power required for all three nozzles.</td>
<td>46</td>
</tr>
<tr>
<td>5.2</td>
<td>Static temperature, maximum model surface temperature, and heating power required for a model placed downstream near the tunnel exit for all three nozzles.</td>
<td>46</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Hypersonic facilities categorized by test duration and operating velocity [9].</td>
</tr>
<tr>
<td>1.2</td>
<td>The pressure vessel.</td>
</tr>
<tr>
<td>1.3</td>
<td>The mechanical valve.</td>
</tr>
<tr>
<td>1.4</td>
<td>The electro-pneumatic vessel.</td>
</tr>
<tr>
<td>1.5</td>
<td>The plenum tank.</td>
</tr>
<tr>
<td>1.6</td>
<td>The planar converging-diverging nozzle.</td>
</tr>
<tr>
<td>1.7</td>
<td>The 4.8 x 4.8 inch test section.</td>
</tr>
<tr>
<td>1.8</td>
<td>The second downstream throat.</td>
</tr>
<tr>
<td>1.9</td>
<td>The entire supersonic wind tunnel without the diffuser</td>
</tr>
<tr>
<td>1.10</td>
<td>Computer model of Cal Poly’s supersonic wind tunnel</td>
</tr>
<tr>
<td>2.1</td>
<td>State diagram schematic of the supersonic wind tunnel.</td>
</tr>
<tr>
<td>2.2</td>
<td>T-s Diagram of the supersonic wind tunnel.</td>
</tr>
<tr>
<td>3.1</td>
<td>Temperature properties for the tunnel with a nozzle width of 1 inch.</td>
</tr>
<tr>
<td>3.2</td>
<td>Plots of the temperature properties for the tunnel at the lowest possible nozzle throat width.</td>
</tr>
<tr>
<td>3.3</td>
<td>Temperature properties for the tunnel at a test section Mach number of 8.</td>
</tr>
<tr>
<td>4.1</td>
<td>Initial mesh used to numerically verify flow behavior within the wind tunnel.</td>
</tr>
<tr>
<td>4.2</td>
<td>ICEM model with increasingly more coarse meshes used to test model convergence.</td>
</tr>
</tbody>
</table>
Hypersonic flows contain many unique attributes that differ them from typical supersonic flows. Some of these phenomena include shock and boundary-layer interactions, entropy layers, large viscous dissipation and viscous interactions, chemically reacting flow, and non-equilibrium effects [1]. Many of these properties can have critical consequences on the design and implementation of aircraft and spacecraft. To ensure that these effects are accurately modeled and that any undesired hypersonic effects during flight are avoided and accounted for, testing and verification procedures are imperative.

There is no single set of conditions that define hypersonic flow and the characterization of a flow as hypersonic depends heavily on the extent that the phenomena listed above affect a model or vehicle. However, most modern hypersonic flight can occur across Mach numbers of 3 to 35 or higher [1]. These speeds, often requiring flight at the upper atmosphere or reentry from orbit to achieve, make it expensive and difficult to organize hypersonic flight tests. Therefore, more accessible alternatives in the form of ground testing facilities constitute many of the hypersonic testing applications done in the United States and the world today.

Unfortunately, ground testing requires many compromises on the testing conditions to model hypersonic flight. With ground-level air conditions having much more pressure and density than that at high altitudes, not all of the air properties and hypersonic phenomena experienced during flight can be modeled on the ground. As a result, there exists a large variety of hypersonic ground facilities, most in the form of wind
tunnels, that are each built to model a unique, desired aspect of hypersonic flight. To expand the scope of Cal Poly’s aerospace testing capabilities and to model a hypersonic testing regime that would proliferate research opportunities, the Cal Poly Aerospace Engineering Department seeks to develop a hypersonic wind tunnel.

1.1 Existing Hypersonic Ground Testing Facilities and Applications

As with any field of science, engineering, and aerodynamics, experimental testing is critical to validate analytical and numerical models. Additionally, with hypersonics, experimental models are often one of the only ways to simulate the flowfield around aircraft and to investigate the unique physics involved [9]. There are many ways to collect experimental data from hypersonic environments, the most popular of which are flight tests and ground tests. Ground testing in particular provides many benefits including the ability to create controlled, sub-scale conditions with the use of wind tunnels, to rapidly prototype and test new models, and to reduce the cost and time it takes to validate other models as compared with flight testing. However, without the opportunity to fly full-scale aircraft or components in their complete intended flight conditions, hypersonic test engineers need to utilize or design ground testing facilities specifically built to capture only the most important aspects of flight needed to validate designs or models. Additionally, power restrictions force many hypersonic ground facilities to create hard limitations on the test duration, test Mach number, or total achievable enthalpy [9].

These limitations on test duration and total enthalpy have led to the development of a large number of diverse categories of hypersonic facilities. Some of the most popular categories, and their respective operating stagnation enthalpies and test durations, are given in figure 1.1. Generally, wind tunnels with longer test durations cannot
operate with large amounts of stagnation enthalpy and tunnels that generate high amounts stagnation enthalpy require quick run-times. Therefore, all of the categories can be divided into two distinct groups based on the test applications each is suited best for: low-enthalpy facilities and high-enthalpy facilities.

1.1.1 Low-Enthalpy Applications

Low-enthalpy hypersonic flow represents flow at a low enough speed and temperature where the fluid can still be assumed to be calorically or thermally perfect [11]. At this regime, full-scale ground testing is not feasible without an extremely large amount of power. For example, a 5m x 5m test-section blowdown facility requires 700 MW of power to create a flow simulating flight conditions at 35 km of altitude [9]. However, the thermally perfect gas assumption allows these facilities to suit very well for scaled-model applications.

Two critical applications to aerospace analysis and design that can be accurately described and observed at the low-enthalpy assumption are laminar boundary layer
thickness growth and viscous interactions. In 1988, Hornung found that thermally 
perfect hypersonic flows could be dimensionalized, assuming the same gas, orientation, 
and flow quality, such that any dimensional quantity would be a factor of freestream 
Mach number, Reynolds number, and wall temperature alone. [11]. Therefore, low-
enthalpy facilities can be great testing apparatuses for modeling hypersonic effects 
that are directly related by the Mach number, Reynolds number, and/or the wall 
temperature ratio. For example, Ginoux gives a reduced form of the laminar boundary 
layer thickness equation for a flat plate as,

\[
\frac{\delta}{x} = \sqrt{\frac{C^*}{Re_{\infty, x}}} \left( 3.07 + \frac{0.58(\gamma - 1)}{2} M_\infty^2 + \frac{1.93 T_w}{T_\infty} \right) \tag{1.1}
\]

where \( \delta \) is the boundary layer thickness and \( C^* \) is the Chapman-Rubeshin factor [8]. 
As long as the tunnel is sized to operate at the appropriate Mach number, the model 
is sized to create the appropriate Reynolds number, and the model is heated or cooled 
to match the wall temperature ratio that would be seen in flight, the model will be 
accurate to the full-scale boundary layer thickness.

Understanding the thickness of the laminar boundary layer in hypersonic flight and 
design is critical. As the Mach number increases, the thickness of the boundary layer 
increases and may become large enough to interact with other surface effects. For 
instance, shock interactions, or interactions between the shock and boundary layer, 
are important to accurately model to prevent unwanted separation [12]. At high Mach 
numbers, additional layers added to the boundary layer, such as the entropy layer, 
add more complexity to these shock interactions. The viscous interaction with the 
entropy layer can be modeled by the equation,
\[ \bar{\chi} = \frac{M^3}{\sqrt{Re_{\infty,x}} \sqrt{C_{\infty}}} \tag{1.2} \]

where \( \bar{\chi} \) is the laminar viscous interaction parameter and \( C_{\infty} \) is the Chapman-Rubesin factor that varies with wall temperature ratio [9].

In other applications where most of the properties cannot be directly scaled, ground testing can be used to provide a generic approximation of how a system would react under various hypersonic flow conditions. Manipulating test section temperature and pressure within a supersonic wind tunnel could allow one to visualize the reaction a material or body may have to the temperatures and pressures seen during flight. For example, sublimation, a key component of material ablation during hypersonic flight, is directly caused by high temperatures and low freestream pressures [13]. Carefully selecting a material that can sublimate within controlled temperatures and pressures in a low-enthalpy facility can allow one to visualize and observe the basic effects of ablation on aerodynamics and system integrity.

While low-enthalpy facilities can take advantage of scaling to meet Mach number and Reynolds number requirements to simulate full-scale flight, both parameters are coupled. Assuming a constant Mach number, the Reynolds number is affected by the pressure and temperature of the test section, which in turn is dependent on the total pressure and temperature of the facility. Therefore, any facility that will be built with the intention to replicate low-enthalpy hypersonic flight needs to be carefully designed to handle pressures and temperatures to get the correct Mach and Reynolds numbers.
1.1.2 Low-Enthalpy Facilities

One of the most popular facilities used to generate low-enthalpy supersonic and hypersonic flow is the intermittent blowdown wind tunnel. Air is pressurized upstream in a tank or other storage facility and separated from a converging-diverging nozzle with a valve or diaphragm arrangement [15]. Blowdown tunnels often produce high-pressure and high-Reynolds number flow over a possible large span of Mach numbers. While the tunnel cannot operate continuously due to the limited amount of air that can be stored upstream, it tends to run for longer than other hypersonic facilities. In addition to the large range of Mach numbers and test durations, blowdown tunnels also have the benefits of being relatively cheap to construct, not needing excess power to start the tunnel, and requiring relatively fast starts that reduce the load on a model and the risk of damage to the facility upon failure. One of the major downsides to using such a facility for hypersonic applications is that, while the flow can be heated in the form of flow-through or batch heaters, the amount of enthalpy generated is very limited. Apart from the low-enthalpy applications used above, other high-enthalpy facilities are typically used to replicate hypersonic flight and testing applications.

A specialized type of blowdown tunnel is the Ludweig tube which uses a long tube with a diaphragm instead of a tank as a high-pressure reservoir. Supersonic flow conditions are created as the diaphragm ruptures and expansion waves are reflected across the tube. While a weak shock will propagate to the test section, expansion waves will travel in the opposite direction until it reaches the end of the long driver tube. At that point, it will reflect and travel back to the test section. Supersonic flow will last as long as the expansion wave is still traveling through the driver tube. This allows the upstream pressure to remain constant as opposed to that of a traditional blowdown tunnel at the cost of a typically lower test duration.
Another specialized low-enthalpy facility is the gun tunnel. As opposed to valves or diaphragms, the main driving force for starting this tunnel is a piston configuration. As the piston fires, the working fluid isentropically compresses to a pressure that can reach around 30 atm. The fluid then flows through a nozzle generating short-term hypersonic flow. The high pressures allow the tunnel to generate the very large pressure ratios required to operate flow at very high Mach numbers. While offering a nominal test duration one magnitude higher than similarly operating high-enthalpy shock tunnels, it still remains on the order of milliseconds. Additionally, the piston tends to create shock reflections that can be difficult to manage.

1.1.3 High-Enthalpy Applications

At high enthalpy, real-gas effects become critical. When a bluff body enters hypersonic flow, a strong shock forms near the model nose causing vibrational excitation and chemical reactions. Low-enthalpy, thermally-perfect fluid molecules tend to have two major categories of motion: translational and rotational. The shape of the molecule will determine its degrees of freedom, as well as the magnitude of kinetic energy, for each category. Translational kinetic energy is the easiest to measure, so that is the property that is averaged to find temperature. However, as a molecule gains enough kinetic energy, a significant portion of energy is added to a third category: vibrational motion. This leads to a fluid having more internal energy than a thermally perfect gas at the same temperature. As such, a thermally perfect gas assumption can no longer apply. Additionally, as more energy is added, the vibrational motion may be so great that the molecular bonds may fail and the molecule will split into atoms. As the molecules dissociate and recombine in a fluid, the chemical energy released further increases the internal energy. It is important for any model representing flow at these enthalpy levels to consider these effects.
Molecular dissociation becomes dominant over a large portion of the hypersonic regime. In these cases where dissociation is dominant, the binary scaling law can be applied to the model flow to preserve any non-equilibrium chemical effects [4]. Anderson gives the binary scaling law as,

$$u \left( \frac{dc_O}{dx} \right) + v \left( \frac{dc_O}{dy} \right) = \frac{M_O}{\rho} k_f \left( \frac{\rho c_O}{M_O} \right) \left( \frac{\rho c_M}{M_M} \right), \quad (1.3)$$

where $c_O$ and $c_{O_2}$ is the mass fraction of $O$ and $O_2$, $u$ and $v$ is the flow velocity in the $x$ and $y$ directions respectively, $M_O$, $M_{O_2}$, and $M_M$ are the molar masses of $O$, $O_2$, and the collision partner, $M$, $\rho$ is the flow density, and $k_f$ is the forward reaction rate constant described by the Arrhenius equation [1]. Additionally, from Hornung, the rate of dissociation for an ideal chemically reacting gas is,

$$R_D = \rho T^n e^{-\frac{D}{kT}} (1 - \alpha), \quad (1.4)$$

where $R_D$ is the rate of dissociation, $\rho$ is the gas density, $T$ is the temperature, $n$ is a dimensionless constant, $D$ is the dissociation constant, $k$ is the Boltzmann’s constant, and $\alpha$ is the mass fraction of the dissociated gas [11]. Combining the equations and using the non-dimensional variables, $x' = \frac{x}{L}$, $y' = \frac{y}{L}$, $u' = \frac{u}{U_\infty}$, $v' = \frac{v}{U_\infty}$, and $\rho' = \frac{\rho}{\rho_\infty}$, gets,

$$\frac{L}{U_\infty} R_D = u' \left( \frac{dc_O}{dx'} \right) + v' \left( \frac{dc_O}{dy'} \right) = \frac{\rho_\infty L}{U_\infty} \rho' T^n e^{-\frac{D}{kT}} (1 - c_O). \quad (1.5)$$

From this equation, Hornung found that for chemically-reacting blunt-body flows, any flow parameter will be a function of freestream velocity, freestream pressure, model characteristic length, the mass fraction of the freestream gas, and the wall
temperature [11]. For high-enthalpy bluff body applications, these are the factors that should be focused on when developing a facility.

1.1.4 High-Enthalpy Facilities

As the flow reaches enthalpies where vibrational excitation and chemical reaction of the fluid occur prolifically after strong shocks, freestream density and velocity become much more important factors to replicate than the Mach number [9]. These types of facilities often require higher pressure and temperature reservoirs than that typically found in low-enthalpy facilities. A popular way of generating these high pressures is to use a strong shock wave that propagates through the tunnel at high speeds. Additionally, a popular way to increase the temperature and velocity of the flow is to change the working gas from air to a lighter substance. For example, hydrogen and helium can be heated up to 800K using an electric resistance heater [10]

However, temperatures higher than 800K are often necessary and other methods of heating are needed. One method is to use an electric discharge. These electric arc drivers are able to heat helium up to 20,000K. However, they are mostly used in more traditional continuous or blowdown tunnels and therefore tend to operate at much lower pressures than that required for high-enthalpy testing. Another method is to use a piston configuration similar to that of a gun tunnel. The fluid kinetic energy can be increased through isentropic compression, by the heat released by the combustion needed to drive the piston, or both. This process allows helium to reach a temperature of 2500-4000K while also maintaining the required high pressures.

There are many ground facilities that can create enough enthalpy to replicate many hypersonic and high-enthalpy phenomena. However, some high-enthalpy effects are nearly impossible to replicate in wind tunnels. When the working fluid is heated up
rapidly, molecular energy splits between separate vibrational, rotational, and translational degrees of freedom. Allowing the flow sufficient time to settle allows these energy states to redistribute into a form of equilibrium. However, most wind tunnels do not operate for long enough test durations for flow within the test section to be considered to be at equilibrium. Additionally, non-equilibrium effects can also occur with chemical energy, as dissociated molecules may not have enough time in a wind tunnel test section to return to equilibrium as well. In hypersonic flight, the freestream air is typically very close to equilibrium conditions [9]. Despite these limitations, many hypersonic and hypervelocity facilities have been developed to capture most of the phenomena listed above.

One way to achieve high enough temperature and pressure reservoir conditions is to use a strong shock and a small converging-diverging nozzle. The tunnel can be configured so that this shock reflects in a way that high-temperature, high-pressure, stagnated flow is created at the nozzle inlet. These types of tunnels are known as shock tunnels. The shock can be created by rupturing a diaphragm and allowing a driver gas to expand along a narrow tube. The test duration of a shock tunnel is often dependent on the time it takes for the driver gas to reach the nozzle, which can be on the order of milliseconds. These tunnels can also be fitted with electrical resistance heaters, arc heaters, pistons (known as Stalker tunnels), or a detonation procedure to drive the stagnation temperature even higher.

However, hypersonic flow can reach very high speeds and enthalpies, to such a degree that even shock tunnels cannot reach the flow conditions desired. In that case, it is necessary to not stagnate the gas before it enters the nozzle and test section. One way to achieve this is to add a second tube, known as an acceleration tube, separated by a weak diaphragm to the end of the shock tube. The acceleration tube is filled with the driver gas at very low pressures. As the strong shock reaches the acceleration tube,
the second diaphragm is ruptured and the gas within the tube undergoes unsteady expansion. The strong shock combines with the unsteady expansion creates a higher temperature and pressure than what would otherwise be achieved with a stagnated fluid. These tunnels are called expansion tunnels and operate at much shorter test durations than reflected shock tunnels to generate much higher pressures and temperatures. However, expansion tunnels are not always used for high-temperature, high-pressure flow, and have seen applications in combustion and the study of non-equilibrium effects at lower enthalpies.

1.1.5 The Need for Additional Facilities

Unfortunately, hypersonic facilities are limited in scope. Many of the tunnels listed above are confined to a few research laboratories and research to be done by professors, companies, and students requires reservations, approval, and travel to these complexes. Additionally, hypersonic flow applications, re-entry in particular, are very tough to model analytically or computationally. Finding ways to make hypersonic testing available and accessible can help bring a new spark to discovery and innovation in a notoriously complicated field.

One critical challenge that affects the design of all hypersonic vehicles is that of avoiding excess material ablation. High-enthalpy ablation is comprised of many complex processes including high-temperature non-equilibrium chemistry, radiation, turbulence, mass/heat transfer, radiation, gas-solid reactions, and mechanical erosion [16]. As a result, ablation modeling is a field of very high interest. However, a cheap and useful way to approximate this effect is to use a sublimating model and compare the magnitude of gas production and transport of material to the boundary layer. This is a field that can be easily tested by adding a heater to a pre-existing supersonic blowdown facility. This thesis explores the viability of installing a heater into the
Supersonic Wind Tunnel at California Polytechnic State University, San Luis Obispo (Cal Poly) in order to stimulate the growth of hypersonic testing at the university and promote the expansion of hypersonic testing opportunities worldwide.

1.2 The Supersonic Wind Tunnel at Cal Poly

Cal Poly’s supersonic wind tunnel is a two-dimensional rectangular intermittent blowdown facility with a variable converging-diverging nozzle. Images of each of its components are given in figures 1.2 to 1.9. It was donated by Boeing in 2000 and represents a $\frac{1}{10}$ scaled-down version of a similar facility designed at Boeing [2]. As it performs currently, Cal Poly’s supersonic wind tunnel (SSWT) is 18.1 ft long with a 4.8 by 4.8 inch test section and a Mach number range of 1.2 to 4. It uses a roughly 1025 ft$^3$ cylindrical pressure vessel (7 ft diameter, 22 ft cylindrical length, 25 ft total length) to store air to a maximum total pressure of 142 psia (9.65 atm) that will discharge to atmospheric pressure through the wind tunnel for a runtime of 7 to 46 seconds [18]. An image of this pressure vessel is given in figure 1.2.

![Figure 1.2: The pressure vessel.](image)

The tunnel contains both a mechanical valve (figure 1.3) and an electro-pneumatic valve (figure 1.4) that is together responsible for running the facility when opened.
A regulator reduces the stagnation pressure to a controlled value in the plenum tank, shown in figure 1.5, where the air stagnates. The plenum tank contains screens and honeycomb meshes that break down large vortices and induce the flow to be more uniform. The pressure in the plenum drives the stagnation pressure and pressure ratio required to operate the tunnel.

![Figure 1.3: The mechanical valve.](image)

![Figure 1.4: The electro-pneumatic vessel.](image)

![Figure 1.5: The plenum tank.](image)

The tunnel contains a semi-flexible planar converging-diverging nozzle upstream of the test section with a minimum throat size of 1 inch given in figure 1.6. The flow then continues into the test section (figure 1.7). A second variable contour throat downstream of the test section (figure 1.8) alters the strength and location of the
diffusing normal shock. Finally, the flow enters the diffuser where it crosses a normal shock and ejects at a subsonic velocity.

Figure 1.6: The planar converging-diverging nozzle.

Figure 1.7: The 4.8 x 4.8 inch test section.

Figure 1.8: The second downstream throat.

The full supersonic wind tunnel without the diffuser can be seen in figure 1.9. A computer model of the facility as it looks at Cal Poly with the diffuser installed is shown in figure 1.10.
Figure 1.9: The entire supersonic wind tunnel without the diffuser

Figure 1.10: Computer model of Cal Poly’s supersonic wind tunnel
2.1 Selecting a Temperature Range

As the pressure tank is outside the facility, it is exposed to weather conditions that can alter the temperature and stagnation properties of the flow as it enters the settling chamber. The latter alteration is important in that any appreciable differences in the temperature of the air within the reservoir, depending on location, will create a non-uniform temperature. This will make a single stagnation temperature hard to determine to calculate flow properties. For this reason, 7075-T6 aluminum wedges comprise 70% of the volume of the tank to ensure that the air temperature remains uniform at every location. However, this massive volume of aluminum prevents more air from entering the tank and shortens the test duration for a given choked mass flow rate. By adding a heater and temperature controller downstream, the air temperature can remain uniform without the need of the wedges.

The next consideration that needs to be made is the issue of avoiding condensation and liquefaction. Condensation of water vapor can be prevented by ensuring that the relative humidity of the flow anywhere in the tunnel does not meet or exceed a value of one. The equation for relative humidity is given below,

\[ RH = \frac{P_{H_2O}}{P_{vH_2O}^*}, \quad (2.1) \]

where RH is the relative humidity, \( P_{H_2O} \) is the partial pressure of water in air, and \( P_{vH_2O}^* \) is the vapor pressure of water at equilibrium. However, the temperature in the
facility can reach such low values and can drive the vapor pressure of water to very low values, even after heating the air, that the only way to keep the relative humidity below unity is to remove water content from the air. This can be done through the use of a dryer and can easily be installed before the plenum if needed.

Liquefaction of air, or the change of phase of Nitrogen, \(N_2\), or Oxygen, \(O_2\) to a liquid state, cannot be avoided with the use of a dryer. As a first estimate, the relation given by Pope [17] can be used to determine the onset of liquefaction of air. That equation is given below,

\[
\log_{10}(P_{\text{liq}}) = \left(\frac{-605.4}{T} + 4.114\right),
\]

where \(P_{\text{liq}}\) is the pressure of liquefaction in atm and \(T\) is the temperature in °R. As long as the pressure at any location is higher than the pressure of liquefaction at that location’s temperature, liquefaction can be avoided.

In a blowdown supersonic wind tunnel, the flow needs to enter the diffuser and eject at subsonic speeds. Therefore, there needs to be a normal shock at some point after the test section. Since pressure and temperature increase significantly after a normal shock and flow velocity is fastest in the test section, the pressure and temperature both reach their minimums in the test section. Therefore, to avoid condensation and liquefaction at any point in the wind tunnel, it is only necessary to prevent both from occurring within the test section.

However, the maximum stagnation temperature of the facility is limited by that which the walls of the facility can handle without melting. The walls of the facility are composed of 7075-T6 aluminum and there is currently no insulation system in
place to keep the walls from melting. Therefore, the total temperature should not exceed the melting point of aluminum at 890°F (750K).

The condensation/liquefaction temperature and maximum reservoir stagnation temperature give limitations to the temperature and Mach numbers that the system can operate at. However, a desired operating temperature and Mach number cannot be selected without a specific test procedure in mind. Different test procedures will vary based on the needs of those conducting the test. However, since the flow enthalpy can be increased, it is of interest to see if this facility can expand to be able to be used for hypersonic testing. One plausible application of hypersonic testing that may be able to be achieved in this updated facility is the study of ablation and sublimation of bluff bodies re-entering Earth’s atmosphere at hypersonic speeds [3].

2.2 Modeling Sublimation

For low-enthalpy applications, materials chosen are often ones with a vapor pressure near typical test section pressures and at temperatures near or below room temperature. Some examples of historically used materials include camphor, CO2 (dry ice), water ice, naphthalene, ammonium chloride, and wax [13]. However, naphthalene and camphor have grown to be more popular than the rest due to their highly robust and affordable manufacturability. Naphthalene has a melting temperature of 353 K making casing very easy to accomplish with the material, in addition to its ability to be milled and seared. Additionally, it emits photovoltaic waves when disturbed allowing for some new flow visualization options. Camphor is a very similar material with a higher melting temperature. This makes it more suited for shape change studies.
2.3 Thermodynamic Model

In order to find the power required to run the heating unit and the liquefaction onset within the test section, the wind tunnel needs to be split into multiple thermodynamic stages. A 2-D schematic of the tunnel divided into thermodynamic stages is given in figure 2.1 while a T-s diagram of the wind tunnel is given in figure 2.2. Using the equations and formulas given through the rest of this section, the pressure, temperature, and specific entropy of the air at each of the stages can be found.

![Figure 2.1: State diagram schematic of the supersonic wind tunnel.](image)

The first stage consists of the air tank where the stagnation temperature can be read. Since aluminum sheets acting as thermal loads exist to keep the tank at a constant temperature during blowdown, a value of 70°F is used in the following analysis. The air within the tunnel can be assumed to behave as an ideal gas and the flow can be assumed to be calorically perfect with a specific heat ratio of $\gamma = 1.4$. 

19
Figure 2.2: T-s Diagram of the supersonic wind tunnel.

After the pressure tank, the flow can be assumed to remain isotropic until it enters the regulator. The regulator acts as a throttle reducing the stagnation pressure from that in the tank to a controlled amount that drives the tunnel. The throttling process can be modeled as adiabatic and the stagnation temperature will not change across the regulator. This pressure reduction can be controlled to drive supersonic flow through the test section. The ideal value of the reduced stagnation pressure can be found by setting the pressure after a normal shock at the tunnel exit, $P_2$, equal to the back pressure. In this case, the back pressure is equal to the atmospheric pressure of 14.7 psia (101.3 kPa). With a given test section Mach number, the pressure ratio across a normal shock is found using equation 2.3,

$$
\frac{P_2}{P_1} = 1 + 2\frac{\gamma}{\gamma + 1} (M_t^2 - 1),
$$

(2.3)
where the subscripts 1 and 2 refer to the properties before and after the normal shock respectively. From there, the ideal stagnation pressure after the regulator at state 2, $P_{01}$, can be found with equation 2.4,

$$\frac{P_0}{P} = \left[1 + \frac{\gamma - 1}{2} M^2\right]^{(\frac{\gamma}{\gamma - 1})}, \quad (2.4)$$

the isentropic flow relation for pressure. In this case, $P_0 = P_{01}$, $P = P_1$, and $M = M_t$.

The subsequent relation for temperature is given by equation 2.5,

$$\frac{T_0}{T} = 1 + \frac{\gamma - 1}{2} M^2, \quad (2.5)$$

The test section Mach number determined by the cross-sectional area ratio between the test section and nozzle throat by equation 2.6,

$$\left(\frac{A}{A^*}\right)^2 = \frac{1}{M^2} \left[\frac{2}{\gamma + 1} \left(1 + \frac{\gamma - 1}{2} M^2\right)\right]^{\frac{\gamma + 1}{\gamma - 1}}, \quad (2.6)$$

where $M = M_t$ in this case. Afterward, the flow enters the Plenum chamber isentropically. The heater will be added to the Plenum tank. Since the air flowing through the Plenum chamber is heated, and the Plenum is short enough that friction can be ignored, this stage of the wind tunnel was analyzed using a Rayleigh flow, or constant-area flow with heat addition and without friction, method. As heat is added to a compressible internal flow, the flow will tend towards a value with a Mach number of 1. Using this reference point, the temperature, pressure, and Mach number of the flow at any point within the Plenum chamber can be related through equations 2.7 and 2.8 [1],

21
\[
\frac{T}{T^*} = \left[\frac{(1 + \gamma)M}{1 + \gamma M^2}\right]^2,
\]  
(2.7)

\[
\frac{P}{P^*} = \frac{(1 + \gamma)}{1 + \gamma M^2}.
\]  
(2.8)

Each equation is used at the beginning and end of the chamber to find changes in temperature and pressure across the heating unit and the air properties in stage 3. Equation 2.6 is used just before the converging-diverging nozzle using the Plenum cross-sectional area to find the Mach number entering the nozzle. The amount of heat added will determine the stagnation temperature entering the nozzle. Afterward, the flow expands isentropically through the nozzle following the isentropic flow equations, equations 2.4 and 2.5, with the new stagnation values to get the air properties in stage 4. The flow reaches a Mach number of 1 in the throat and continues accelerating isentropically to the test section using the same equations to get the air properties in stage 5. The nozzle inlet stagnation properties are needed to find the choked mass flow rate of the wind tunnel.

Finally, friction should not be ignored for the long portion of the wind tunnel from the test section to the tunnel exit or diffuser. As a result, a Fanno flow model, or adiabatic constant-area flow with friction, can be used to find the change of states between these two locations. The adiabatic assumption is made to simplify the analysis and represents a reasonable approximation for blowdown wind tunnels with short test durations such as this one. Friction will also tend the flow towards a Mach number of 1 and an integral relation between the Mach number and non-dimensional tunnel length as a function of the Fanning friction coefficient can be found. Assuming the Fanning friction coefficient remains constant, the relation can be reduced to equation 2.9 [1].
\[
\frac{4fL}{D} = -\frac{1}{\gamma M} - \left[ \left( \frac{\gamma + 1}{2\gamma} \right) \ln \frac{M^2}{1 + \left( \frac{\gamma - 1}{2} \right) M^2} \right] \bigg|_{M_1}^{M_2},
\]  

(2.9)

where \( M_1 \) is the Mach number in the test section and \( M_2 \) is the Mach number before the tunnel exit. Using this equation and equations 2.4 and 2.5, the air properties at stage 6 can be found.

Many of the equations listed above in this section are nonlinear and coupled together. Solving for the temperature and pressure at each of the six stages requires a large amount of iteration. Therefore, it is highly recommended that computational software is used to conduct this analysis.
Chapter 3

ANALYTICAL RESULTS

3.1 Current Nozzle

The heat required from a heater is given by the following equation,

\[
\dot{Q} = \dot{m}_{\text{choked}} c_p (T_{0(\text{nozzle})} - T_{0(\text{tank})}).
\]  

(3.1)

The stagnation temperature in the tank is given as the atmospheric temperature. The stagnation temperature in the nozzle is the design point of the facility as described in the above section and is limited by the maximum temperature that the walls of the facility can handle before melting. This maximum temperature of 750 K is the temperature that will be considered for the results. This allows for the amount of power required for the heater to be dictated by the choked mass flow rate, \( \dot{m}_{\text{choked}} \).

Setting the nozzle throat area and test section Mach number constant, the test section properties were found after adding different amounts of heat power in the Plenum section during blowdown operation. The amount of heat added to the flow directly affects the choked mass flow rate of the tunnel. The choked mass flow rate can be found by analyzing the conditions at the nozzle throat where the Mach number can be assumed to be equal to one. The equation for choked mass flow rate as analyzed at the throat is given as

\[
\dot{m}_{\text{choked}} = \rho A M \sqrt{\gamma R_{\text{Air}} T} = \rho^* A^* M^* \sqrt{\gamma R_{\text{Air}} T^*}.
\]

(3.2)
The values for $\rho^*$ and $T^*$ are directly correlated to the stagnation conditions and $M^*$ equals 1. The amount of heat one adds to the flow will dictate how much the stagnation density and temperature will change in the throat and the test section. By extension, this will dictate how much the mass flow rate will change. As a result, changing the mass flow rate as an independent variable will allow the direct calculation of how much heat needs to be added to the flow.

While the converging-diverging nozzle currently installed in Cal Poly’s supersonic wind tunnel can vary in area, it is set to a width of 1 inch to grant a test section area ratio of 4.8. The test section Mach number can be calculated from this area ratio using equation 2.6 to get a test section Mach number of 3.1315.

The stagnation pressure required to drive the flow is chosen to be a value in which a normal shock will occur between the test section and the diffuser/ tunnel exit. This way, the flow can remain supersonic within the test section but return to a subsonic velocity so that the increasing area of a diffuser can serve to decelerate rather than accelerate the flow. This value can be found by calculating the stagnation pressure ratio across the normal shock at the test section Mach number, $M_t$, and the isentropic pressure ratio at the tunnel exit. Currently, the tunnel has no diffuser attached and the tunnel exit area is roughly the same as the test section area. This means that the tunnel back pressure, $P_e$, is equal to the static pressure after the normal shock. Additionally, the stagnation pressure after the normal shock, $P_{02}$, is equal to the exit stagnation pressure, $P_{0e}$. The wind tunnel pressure ratio, $\frac{P_0}{P_e}$, can be found by multiplying the ratios, $\frac{P_{01}}{P_{02}} \frac{P_{0e}}{P_e}$. Understanding that the tunnel back pressure is equal to the atmospheric pressure of roughly 14.7 atm or 101.3 kPa, the stagnation pressure required to position the normal shock can be found appropriately.
\[ P_0 = \frac{P_{01}}{P_{02}} \frac{P_{0e}}{P_e} \]  \hspace{1cm} (3.3)

The two pressure ratios can be found using the following equations,

\[ M_{t2}^2 = 1 + \frac{[(\gamma - 1)/2] M_t^2}{\gamma M_t^2 - (\gamma - 1)/2} \]  \hspace{1cm} (3.4)

\[ \frac{P_{02}}{P_2} = \frac{P_{0e}}{P_e} = 1 + \frac{\gamma - 1}{2} M_{t2}^2 \]  \hspace{1cm} (3.5)

where the subscripts 1 and 2 refer to the properties before and after the normal shock and \( M_{t2} \) is the Mach number after the normal shock. For air at \( \gamma = 1.4 \) and a test section Mach number of \( M_t = 3.1315 \), the required stagnation pressure, \( P_0 \) is 67 psia.

Engineering Equation Solver (EES) was used to construct the wind tunnel thermodynamic model and iteratively conduct the analysis. It was found that the variable that needed to be changed to make the model converge was the Mach number in the Plenum chamber before heating without friction. This value is also directly related to the mass flow rate through equation 3.2 since the plenum cross-sectional area and stagnation conditions in the air tank are also known. Changing this value allows for a range of heating power to converge and for the test section properties to be analyzed at that range.

The test section stagnation temperature is plotted against the heating power required to reach that temperature below in figure 3.1a. The dotted line indicates the location at which the stagnation temperature reaches the limits of that which can be handled by the facility. In order to reach this temperature, 960 kW of power is required to be fed into the heater. As indicated in section 2.1, the power available to the
facility currently is a 110 W, 20 A wall outlet. A 1 MW generator would need to be purchased and installed near the facility to reach this amount. Using the power available through the wall outlet, the stagnation temperature would only increase by a few degrees.

(a) Test section temperatures at various heating power amounts.  
(b) Tunnel exit temperatures at various heating power amounts.

Figure 3.1: Temperature properties for the tunnel with a nozzle width of 1 inch.

It was found through equation 2.8 that the stagnation pressure does decrease with heat addition. However, that change is negligible causing the pressure in the test section to appear to remain constant with different amounts of heat addition. Therefore, adding heat to the flow will only increase the temperature within the test section. Using this knowledge and equation 2.2, the temperature that would theoretically cause liquefaction in the test section can be predicted. In order for the air to liquefy, the test section must generate a pressure below the liquefaction pressure at a given temperature. It was found that at all temperatures, even with no heat added, the air would not liquefy. The condensation that occurs in the wind tunnel currently must only be related to the humidity of the air.

The static temperature before and after a normal shock is also plotted against the amount of heating power added to the system further downstream the tunnel in figure 3.1b. The distance between the end of the nozzle and the diffuser is long and
adiabatic flow without friction can be used to find new flow properties before the
diffuser. The Fanning friction factor, $f$, is assumed to be constant along the length of
the tunnel and estimated to be 0.005 [1]. While the Mach number decreases, the static
temperature also increases significantly if high-temperature supersonic conditions are
desired. However, any testing downstream of the test section would require the design
and insertion of a downstream test stand.

### 3.2 Decreased Throat Area

The results for the previous section produce very promising temperature and pressure
results for the goal of heating the flow. The static temperature before any normal
shock is close to the atmospheric temperature and the temperature after the shock
is close to the maximum stagnation temperature at maximum heating. However,
the power required to achieve that flow is incredibly large and requires the purchase
and installation of a 1 MW generator; a cost that can run up to $50,000. Equation
3.1 indicates that the power can be reduced by reducing the choked mass flow rate.
Furthermore, the choked mass flow rate can be reduced by reducing the throat area
according to equation 3.2. Finally, equation 2.6 indicates that the test section Mach
number will increase by decreasing the throat area and keeping the test section area
constant. The Mach number increase can serve to further the goal of developing
a hypersonic facility with the benefit of additionally possibly decreasing the power
required to heat the flow. However, an increased test section Mach number also
requires a larger wind tunnel pressure ratio, leading to either an increase in tank
stagnation pressure or a decrease in back pressure. Increasing the tank stagnation
pressure will also increase the mass flow rate and increase the power required to heat
the flow. The following analysis will serve to determine if decreasing the throat area
will decrease the overall power requirement.
The variable-area nozzle can create a maximum test section Mach number of $M_t = 4$. Using equation 2.6 the area ratio for a flow at this Mach number can be deduced to be 10.7 and the throat width to be 0.4478 in. The stagnation pressure required to drive this flow needs to be increased to drive a normal shock past the test section and model. Using equations 3.3 to 3.5, the required stagnation pressure is found to be about 120 psia. This pressure is near the limit pressure that can be handled by the tank at Cal Poly and represents a second limit to the test section Mach number along with the current nozzle geometry.

Using Engineering Equation Solver (EES), the test section stagnation temperature is plotted against the heating power required to reach that temperature for the Mach 4 case below in figure 3.2a. The dotted line indicates the location at which the stagnation temperature reaches the limits of that which can be handled by the facility. Only 430 kW of heating power is required to reach this temperature, as opposed to the 960 kW required for the Mach 3.1 case. While this is a large improvement, a generator would still need to be purchased and installed to accommodate the 430 kW of heating power. Additionally, the test section static temperature before and after a normal shock is included in this figure. The static temperature before the normal shock is around 250 K while the static temperature after the normal shock is near the stagnation temperature at 715 K.

Equation 2.2 was used again to predict the temperature at which liquefaction would occur for the lower pressure at Mach 4 conditions. However, the pressure remains below the liquefaction pressure at all temperatures and liquefaction still does not occur at this Mach number.

The static temperature before and after a normal shock is also plotted against the amount of heating power added to the system further downstream the tunnel in figure 3.2b. As friction from the tunnel walls decreases the momentum of the flow, models
placed in the downstream portion of the test section will encounter higher static temperatures and lower Mach numbers than if placed near the nozzle. The higher static temperatures may be of interest when testing sublimation.

![Figure 3.2: Plots of the temperature properties for the tunnel at the lowest possible nozzle throat width.](image)

(a) Test section temperatures at various heating power amounts.  
(b) Tunnel exit temperatures at various heating power amounts.

The limitations of pressure ratio and nozzle geometry can be overcome by adding a vacuum pump or chamber to the tunnel exit and designing a new isentropic nozzle with a smaller throat. The former will serve to increase the pressure ratio required to drive back a normal shock without increasing the stagnation pressure in the tank while the latter will increase the test section Mach number and further reduce the choked mass flow rate. Since the stagnation pressure does not change, the power required to heat the flow will decrease.

Changing the size of the throat to give a test section Mach number of $M_t = 13$ will reduce the heating power required to the 2 kW that can be provided by the wall outlet. However, the test section pressure and temperature range is much lower. Equation 2.2 predicts that the test section pressure is well above the liquefaction pressure at the range of operating test section temperatures. Any additional heating of the flow
to avoid liquefaction will melt the walls of the facility, rendering flow at this Mach number impractical for Cal Poly’s facility for the time being.

However, a test section Mach number of $M_t = 8$ will allow the flow to avoid liquefaction only when heated to its maximum possible stagnation temperature of 750 K. This trend is shown in figure 3.3c, where the black line represents the liquefaction pressure over the range of operating test section pressures. The pressure only gets below liquefaction temperature at around 50 K, the temperature reached when the stagnation temperature is nearly at its maximum value of 750 K. Therefore, this Mach number represents the effective limit that the facility can handle. The power required by the heater at this temperature is shown in figure 3.3a at a value of 25 kW.

The static temperature before and after a normal shock is also plotted against the amount of heating power added to the system further downstream the tunnel in figure 3.3b.
(a) Test section temperatures at various heating power amounts.  
(b) Tunnel exit temperatures at various heating power amounts.  
(c) Liquefaction onset curve and test section temperature range

Figure 3.3: Temperature properties for the tunnel at a test section Mach number of 8.
Now that an analytical model has been developed and a solution has been reached, it is important to check the validity of this solution using an alternative modeling method. Two popular options for verifying analytical models are numerical and empirical models. Due to a number of assumptions that allow the simplification of the wind tunnel system, limitations with the time frame and equipment to conduct a proper empirical test, and the lack of securely-fastened test equipment to collect usable test data for the current tunnel configuration, a Computational Fluid Dynamics (CFD) software was used as the primary means of verification. CFD is a powerful tool that can be used to model fluid and thermal flow by splitting a problem into a series of finite elements and reducing the governing equations into a system of equations that can be solved using numerical methods. As such, by defining the geometry of the wind tunnel, assigning appropriate boundary and initial conditions, and running an appropriate solver, one can reach a solution without needing to derive and directly solve a large number of physics-based equations. Ansys ICEM CFD software is used to construct and mesh the model while Ansys Fluent is used to construct the solver, run the model, and post-process the results.

4.1 Assumptions

The goal of running a CFD model for the proposed wind tunnel is to prove that the assumptions used to derive the governing equations and results reached with the analytical solution align with those reached by the numerical solver. This means that
the numerical solver needs to model a control case of the tunnel with no heat added, ensuring that the numerical model is valid and that the solver can properly compare with the analytical solution.

One of the downsides of using a numerical method is that, depending on how extensive or complicated the model geometry, boundary conditions, initial conditions, and solver defined are, a large amount of time and computing power may be needed to run a solution. Defining accurate assumptions is again important because, while an accurate solution may still be able to converge without a large number of assumptions, reducing the system to a much simpler one that yields the same significant results can prevent unnecessary wastes of time, money, and work.

Perhaps one of the largest geometric assumptions that can be made is to reduce a three-dimensional geometry into a two-dimensional axisymmetric model. The tunnel is three-dimensional and contains many complicated geometric transitions through the flow. One such example is how the tunnel transitions from a circular to a rectangular cross-section from the plenum chamber to the throat and back to a circular cross-section in the diffuser. However, the goal of the CFD solver is to compare the test section solution at the center of the tunnel, where the flow at edges and transition points are not critical and can be assumed to not propagate down, to the test section. Although the pipes feeding into the plenum chamber twist in three dimensions, the effect this has on flow behavior is small and can be ignored for an initial analysis. Finally, as boundary layer effects do not need to be modeled since boundary layer corrections were not applied to the analytical model, the flow can be assumed to be a direct factor of the area of the cross-section at a given point rather than its specific geometry. As such, an axisymmetric model around the tunnel centerline with an effective radius equal to the square root of the area of the tunnel divided by pi at that point is created.
Blowdown wind tunnels have fairly short test durations allowing for an adiabatic boundary condition to be applied to most of the walls. Additionally, the centerline of the axisymmetric model can be defined with a symmetric boundary condition. In fact, there are only two critical surfaces that define how the tunnel operates and to which special boundary conditions should be applied. The first is the tunnel exit, which is set to the pressure output boundary condition. Treating the exit as a pump sucking the high pressure tank air out of the tunnel is effectively the same as high pressure air blowing out of the tunnel to atmospheric pressure. The second boundary condition is the walls of the plenum chamber, which are modeled using a constant heat flux. While the heater will likely contain additional heating elements that add power away from the walls and towards the center of the chamber, the chamber air can be assumed to be static and the method of heat addition can be assumed to have a negligible effect on the outcome. The analytical model is assumed to be quasi-steady so any transient or controlled power variation can also be ignored and a constant heat flux can be assumed.

The initial conditions constitute the same initial conditions that would be expected for a full-scale blowdown facility: an initial high pressure located solely in the tank and a lower pressure located in all other spaces within the tunnel. However, that lower pressure cannot be set to atmospheric pressure as the exit boundary condition needs a pressure differential for the simulation to run. Therefore, a pressure slightly higher than that of the atmosphere is set for every part of the tunnel outside of the tank. The two-dimensional heat flux is set to the 1 MW/m derived earlier at the desired test section Mach number of 3.13.
4.2 Constructing the Model

The tunnel geometry and mesh were created using the Ansys ICEM software. See Appendix A for an example of the replay script used to generate the axisymmetric tunnel geometry. The mesh was generated by first creating a rectangular grid that encapsulates the entire model. However, the grid needs to be subdivided and rearranged to fit the surfaces within the geometry that defines the flow. The initial grid is split at critical points that define changes in geometry and parts and these split grids are deleted and combined until they can be fit into surfaces. Then each of the edges are associated to curves where the mesh resolution at that edge gets defined. In the end, a structural mesh capable of running the simulation to reasonable accuracy is generated. This mesh can be seen in figure 4.1.

![Initial mesh used to numerically verify flow behavior within the wind tunnel.](image)

**Figure 4.1:** Initial mesh used to numerically verify flow behavior within the wind tunnel.

One important aspect of any numerical testing is to ensure that a model can converge. To do so, the initial mesh generation must be modified to show that the simulation approaches a solution when run with finer and finer meshes. Therefore, multiple meshes were created with varying resolutions to pass into Fluent. These meshes make up the rest of figure 4.2.

Once the geometry and mesh are constructed, boundary conditions on the curves and surfaces were defined. The boundary conditions used in this program were the same as listed in section 4.1, using the options defined by the ICEM program: ‘symmetric’
for the centerline axis, ‘wall’ for the walls, ‘outlet’ for the outlet, and ‘inlet’ for the top plenum wall.

The final model can then be exported from ICEM CFD to Fluent to generate the solver and run the model. As wind tunnels are well-studied controlled systems, a fairly basic and quick-to-run solver was sufficient for our goals of verifying the analytic results. Therefore, a single-phase inviscid model for an ideal/perfect gas was selected as the solver.

4.3 Running the Simulation

Selecting or building a solver is a critical step in conducting any computational fluid dynamics analysis. The solver is responsible for providing the system of differential equations that will be solved numerically across the geometry, mesh, and boundary conditions defined in the section above. Most of the time, the solver must calculate the system of equations iteratively, where a set of solutions across a single time-step is necessary to approximate a solution at the next future time-step. This is known as a real-time solver, and depending on the complexity of the solver, the computing power required at each time-step, and the total number of time-steps desired to reach the final solution, this process can be very time and cost-consuming. Therefore, it
is imperative to implement enough physical assumptions about the system to reduce the computing time as well as the power required to solve the equations and to use as large of a time-step as possible to lessen the total calculation duration. However, it is important to note that using too large of a time-step may prevent a solution from converging. The time-step should be small enough so that using even smaller time-steps would not create different results.

In creating the thermodynamic model for the wind tunnel, a number of assumptions were utilized for the air and for the flow between numerous different states. To ensure the CFD solution provides an accurate comparison of results, most of those assumptions are taken into account when building the CFD solver. See section 2.3 and figure 2.2 for an overview of those assumptions. The notable exception would be the Fanno flow assumption inside and beyond the test section. Instead, an inviscid solver was used for the entirety of the tunnel, as selecting a viscous solver would drastically increase the solver complexity and solution time. Viscosity is negligible across most of the tunnel leading up to the test section, and comparing the analytical and CFD model at the test section only will be enough to verify the results.

Fluent provides a graphical user interface that separates the solver generation into a number of different tabs split into three different sections. The first section, Setup, contains tabs that help define the physics of the problem. These include General, Models, Materials, Cell Zone Conditions, and Boundary Conditions among others. The second section, Solution, contains tabs related to defining the numerical method used to solve the system of differential equations. This section also allows the user to set the initial conditions and calculate the solution. Finally, the Results section includes all of the post-processing tools including tabs for generating Graphics, Plots, Animations, and Reports among others.
Since the heat flux boundary condition was defined within ICEM, both the non-heating and heating case can be run with the same exact Fluent solver settings. For both cases, after importing the mesh into Fluent and adjusting the scale to correct the units, the solver can be defined to be planar, transient, and density-based with an absolute velocity formation. This is because model uniform flow, the transient starting process, and flow compressibility are respectively desired. The only additional equations not required to conduct a CFD analysis are the energy equations. The basic material properties of air and the aluminum walls of the tunnel are defined and allocated to their respective cell zones in the mesh.

A first-order implicit solver using second-order upwind spacial discretion with a least squares cell based gradient was chosen to define the underlying equations. The other model parameters were kept to the default settings recommended by Fluent.

The problem was initialized as a final step before running the solution. The mesh was first initialized at a room temperature of 300 K and an initial pressure of 200 Pa that will get overwritten. The low initial pressure helps the pressure outlet boundary condition work as intended without creating undesired reversed flow at the tunnel exit. To simulate the pressurized tank, a cell register was created for the area on the mesh that represents the tank. In this case, the tank is a rectangle of length 6.096 m in the x-direction and 0.9271 m in the y-direction with the bottom left corner at \( x = 0 \) and \( y = 0 \). When operating the tunnel, an upstream pressure of 60 psia is required. Converting this to gauge pressure and converting to Pa, the tank register can be patched at a pressure of 312,333 Pa and the rest of the tunnel at the sea-level pressure of 101,325 Pa.

The time-step and number of iterations for each time-step was selected and the calculation was run. Ideally, the time-step should be small enough that a result can converge with a relatively small amount of iterations. It was found in this case that
a fixed time-step of 1 microsecond allowed convergence to occur with less than 20 iterations. The solution ran until the flow could completely blowdown to the tunnel exit, around 10,000 time-steps.

4.4 Results

4.4.1 No Heating

Figures 4.3 through 4.5 below show the results of the simulation for pressure, static temperature, and Mach number distribution across the tunnel.

![Static Pressure](image)

**Figure 4.3: Pressure results of the CFD simulation across the tunnel.**

As expected with any supersonic blowdown facility, the flow accelerates through the converging-diverging nozzle. Figure 4.5 illustrates that the flow is choked. The Mach number continues to increase to its predicted value of 3.1 before encountering a shock and returning to subsonic conditions. The pressure and temperature trends highlight this phenomenon as they both decrease across the throat and diverging section of the nozzle.

Table 4.1 below compares the results of the simulation to that of the analytical calculations. Both the Mach number and the static temperature are very close to each
Figure 4.4: Static temperature results of the CFD simulation across the tunnel.

Figure 4.5: Mach number results of the CFD simulation across the tunnel.

other indicating that the simulation is accurate to the analytical results within a one percent error. The pressure is a little farther off possibly due to the decrease in effective test section area due to the boundary layer.

Table 4.1: Pressure, static temperature, and Mach number comparison between the analytical and CFD results for the non-heated case.

<table>
<thead>
<tr>
<th>Property</th>
<th>Analytical</th>
<th>CFD</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>11 kPa</td>
<td>9.34 kPa</td>
<td>15</td>
</tr>
<tr>
<td>Static Temperature</td>
<td>100 K</td>
<td>99.3 K</td>
<td>0.7</td>
</tr>
<tr>
<td>Mach Number</td>
<td>3.13</td>
<td>3.14</td>
<td>0.32</td>
</tr>
</tbody>
</table>
Scaled residuals were used to help determine if the CFD simulation converged to desired results. In this case, many of the scaled residuals were not able to minimize until after about 1500 iterations. An adaptive time-step was used in order to drastically reduce the simulation time. A large time-step was chosen for the initial blowdown of the tunnel as accuracy would not be needed until the tunnel reached a quasi-steady state. After about 1500 iterations and the quasi-steady condition was reached, the time-step was reduced to get a more accurate solution at the final time-step. The solution is satisfactory because it aligns with most of the assumptions and knowledge of the current facility.

4.4.2 Heating

Figures 4.6 through 4.8 below shows the results of the simulation for pressure, static temperature, and Mach number distribution across the tunnel.

![Figure 4.6: Pressure results of the CFD simulation across the tunnel.](image)

While these results do not appear as continuous as the previous results, likely indicating a convergence issue, they do remain typical as to what the effects of adding a heater to the wind tunnel would be. The heater causes the static temperature to start at a higher value ahead of the nozzle which decreases the static temperature as
flow accelerates through the test section until a normal shock is reached. The high stagnation temperature along the walls of the test section due to the boundary layer is also noticeable. Pressure seems to increase and velocity seems to decrease across the diverging section of the nozzle while remaining supersonic flow throughout the entire exit to the plenum chamber. This is nonphysical and illustrates issues with the results of this simulation.

Table 4.2 below compares the results of the simulation to that of the analytical calculations.
Table 4.2: Pressure, static temperature, and Mach number comparison between the analytical and CFD results for the heated case.

<table>
<thead>
<tr>
<th>Property</th>
<th>Analytical</th>
<th>CFD</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>11 kPa</td>
<td>10.6 kPa</td>
<td>3.6</td>
</tr>
<tr>
<td>Static Temperature</td>
<td>250 K</td>
<td>328 K</td>
<td>31.2</td>
</tr>
<tr>
<td>Mach Number</td>
<td>3.13</td>
<td>4.33</td>
<td>120</td>
</tr>
</tbody>
</table>

Overall, these results show a much larger discrepancy between the analytical and computational methods. Inability of the CFD model to converge is the greatest cause of this difference. Figure 4.9 shows the scaled residuals of the heated simulation. Due to the complexity of adding the heater to the equation, these residuals were difficult to converge and mostly remained greater than 1, indicating a possible lack of solver convergence. A future project of running a longer simulation with smaller time-steps and other complexity-reducing changes may help make the results more accurate.

Figure 4.9: Scaled residuals of the non-heated CFD simulation
This paper sought to size a heater so that it can be installed into Cal Poly’s supersonic wind tunnel and expand the functionality of the facility. Currently, the problems affecting the facility are that condensation exists in the test section and that large amounts of metal in the air tank act as heat loads that reduce the test duration of the tunnel. Additionally, Cal Poly’s Department of Aerospace Engineering is looking to expand the facility to be used for hypersonic applications.

A thermodynamic model of the facility was created in order to analyze these results. The regulator responsible for controlling the tunnel pressure ratio was modeled as an adiabatic throttle, the heater to be added to the Plenum chamber section was modeled using Rayleigh flow, the flow through the converging-diverging nozzle was modeled as isentropic, and the flow from the test section to the tunnel exit or diffuser was modeled using Fanno flow. The stagnation temperature before the heat addition was determined to be atmospheric temperature, the stagnation temperature after heating was limited by the melting temperature of the walls, and the stagnation pressure after the throttle was determined by setting the back pressure equal to the static pressure after a normal shock at the tunnel exit. It was found that the stagnation pressure changes by a negligible amount during heat addition so the stagnation pressure found after throttling was used to determine the normal shock location necessary to run the tunnel.

The results across the different nozzle sizes are summarized in tables 5.1 and 5.2. These results found indicate that, with the nozzle geometric and running conditions
set up currently at Cal Poly’s Supersonic Wind Tunnel, heating the flow up to the
limit stagnation temperature has noticeable effects on the flow. The test section
static temperature increases from roughly 100 K to around 250 K as seen in figure
3.1a. After a normal shock, such as across a bow shock created by a blunt-nosed test
model, the temperature reaches 715 K. Furthermore, downstream towards the tunnel
exit, a temperature of 520 K is reached before a normal shock and increases to 675
K afterward. However, the Mach number decreases to slightly less than 1.5.

Table 5.1: Test section temperature, maximum model surface temperature, and heating power required for all three nozzles.

<table>
<thead>
<tr>
<th>Model for Tank</th>
<th>TS Mach</th>
<th>TS Temp</th>
<th>Max Model Temp</th>
<th>Power</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant Temperature</td>
<td>3.13</td>
<td>250 K</td>
<td>715 K</td>
<td>960 kW</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>175 K</td>
<td>705 K</td>
<td>430 kW</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>60 K</td>
<td>715 K</td>
<td>24 kW</td>
</tr>
</tbody>
</table>

Table 5.2: Static temperature, maximum model surface temperature, and heating power required for a model placed downstream near the tunnel exit for all three nozzles.

<table>
<thead>
<tr>
<th>Model for Tank</th>
<th>TS Mach</th>
<th>TS Temp</th>
<th>Max Model Temp</th>
<th>Power</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant Temperature</td>
<td>3.13</td>
<td>515 K</td>
<td>680 K</td>
<td>960 kW</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>580 K</td>
<td>680 K</td>
<td>430 kW</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>450 K</td>
<td>700 K</td>
<td>24 kW</td>
</tr>
</tbody>
</table>

Using these results, some hypersonic applications can be tested in this relatively low-
velocity facility. One such application may be the testing of sublimation on re-entry
bodies. With re-entry temperatures reaching the order of 10,000 Kelvin, the metal
behind a bow shock may sublimate. Using a material that sublimates at much lower
temperatures, such as camphor or naphthalene, this increase in temperature both in
general and behind the normal shock may be the increase in temperature needed to
structurally and aerodynamically analyze flow across a sublimating structure.

At the test section Mach number of 3.13, it was found that liquefaction would not
occur anywhere in the tunnel, even without heating the flow. Therefore, the conden-
sation that occurs in the test section during operation is a result of humidity from the water content. To avoid this condensation, it is recommended that a dryer is installed to remove the water content in the flow.

However, the major downside to installing a heater to the wind tunnel as it currently runs is that the heater requires a large amount of power. The 960 kW of power required to run the heater at the limit temperature would require the purchase and installation of a 1 MW generator. These generators can run up to $50,000 and can take up the space of an entire building. Significant funding, research regarding regulations and safety for installing large-powered generators, and time would be needed to make this investment.

To reduce the power and cost to install a heater, the test section Mach number needs to be increased. The limitations on pressure ratio and nozzle geometry for the air tank and variable-size converging-diverging nozzle set the maximum Mach number for the current facility at a test section Mach number of 4. The decrease of the throat area required for this increase in Mach number reduces the mass flow rate and, by extension, reduces the power required for the heater. By reducing the throat area by roughly half, the power required to reach the limit temperature also reduced by roughly half to a value of 430 kW. While a large generator would still be needed, this value constitutes a significant improvement from the 960 kW needed for the previous case.

At the test section Mach number of 4, the test section temperature is around 175 K and the tunnel exit temperature is 485 K. Additionally, the Mach number at the tunnel exit is 1.6. These values are lower than that of the current Mach 3.13 facility and may affect the possibility of testing sublimation. The temperatures after a normal shock are 720 K in the test section and 690 K at the tunnel exit. While these values are slightly higher than that of the Mach 3.13 facility, these temperatures would
only occur at the stagnation line directly in front of a blunt re-entry model. The temperature away from this stagnation line would tend towards the static conditions, rendering sublimation modeling less likely to be as accurate. If a large temperature solely concentrated on the nose of a bluff-body model is desired for the model, it is recommended that sublimation is tested in the test section. If a large overall temperature is desired, it is recommended that sublimation is tested near the tunnel exit in this case. However, a new test model stand would need to be constructed for testing models near the tunnel exit.

Similar to the tunnel currently in operation, no liquefaction occurs anywhere in the tunnel at this test section Mach number, even with no heat addition. A dryer would also be needed to remove condensation in the flow. However, the test section temperature is much lower and condensation is more likely to occur than with the tunnel currently in operation. An investment in a stronger dryer is recommended.

To drive the power required for the heater to an easier-to-implement value, the limits of the facility need to be overcome by reducing the throat area of the converging-diverging nozzle. Furthermore, changing the pressure tank to a higher pressure facility or adding either a vacuum pump or a chamber to the tunnel exit is recommended. Adding the vacuum pump or chamber to the tunnel exit would be easier to implement and would have better effects on the power requirement when compared to increasing the stagnation pressure. However, this may serve to reduce the amount of air that can travel through the tunnel and further decrease the test duration.

By changing the test section area ratio of the facility to create a test section Mach number of 13, the power required to heat the facility to the limit temperature is 2 kW, enough for the heater to be powered by a wall outlet. However, liquefaction at this test section Mach number is unavoidable, even after heating the flow. The maximum test section Mach number that can be reached so that liquefaction is avoided when
the tunnel is heated to the limit temperature is a Mach number of 8. This Mach number generates a lower power limit of 25 kW at that maximum temperature, a value that is much lower than the previous power amounts at Mach 3.13 and 4. However, the tunnel must still be heated to this limit number to prevent liquefaction. It is recommended that a Mach number less than 8 is used so that there is a higher factor of safety to avoid melting the walls and liquefaction.

At the Mach number of 8, the test section temperature is 55 K, the tunnel exit temperature is 475 K, the Mach number at the tunnel exit is 1.875, the temperature in the test section after a normal shock is 720 K, and the temperature at the tunnel exit after a normal shock is 750 K. This produces similar results to the Mach number of 4 and the same recommendations apply. A trade study could be conducted to determine the costs and benefits of lowering the heater power and decreasing the test section temperatures to such low values. The result of this trade study would determine which Mach number to implement the heater at.

Further recommendations for improving this sizing methodology include testing condensation and test duration. Regarding the former, equation 2.1 can be used with the test section properties to determine the onset of condensation. The amount of moisture content in the air needs to be approximated but knowledge of this can determine what the size and power of any dryer needs to be to avoid condensation in the test section. Additionally, this could be used as another constraint that would prevent the use of high test section Mach numbers and result in an increase in heater power.

The latter may require a more in-depth model. Without the metal in the tank to maintain an equilibrium stagnation temperature, the stagnation temperature will decrease in the tank as it depressurizes. This may affect both the isentropic assumption made throughout the flow as well as the amount of power required by the heater. It
is recommended that the discharge of the tank be modeled in future studies of heater sizing for wind tunnels.

Additionally accurate results regarding the flow downstream of the test section can be found by using a more accurate result for the Fanning friction factor, $f$. This friction factor varies with the Reynolds number and depends on the regime of the flow. Both these values change as the flow decelerates downstream from the test section. Anderson [1] recommends a value of 0.005 for supersonic applications but this value could be more accurately determined with experimental or numerical methods at each location within the tunnel. Additionally, the value of 0.005 may not be accurate for high-temperature or hypersonic applications. More research is needed to determine the best approximation for this value.

Overall, adding a heater to Cal Poly’s supersonic wind tunnel would allow the tunnel to be used for more hypersonic applications, such as re-entry sublimation testing or high Mach number tests. Installing the heater into the current facility would require a large generator to be purchased and installed. However, driving the Mach number higher would reduce this power requirement at the cost of requiring a new nozzle and equipment, as well as driving down the temperature in the test section.

5.1 Sublimating Material Selection

Both camphor and naphthalene would work as sublimating model materials in this range of stagnation temperature and pressure. Additionally, the power requirements for sublimation would be well under that required for the maximum heat flux. The vapor pressure of Naphthalene at a temperature of 350 K, 668.5 Pa, corresponds to the static pressure found in the test section at that temperature. Therefore, the minimum power requirements would be 200kW for Naphthalene at $M=3.13$, 75kW for
M=4, and 5kW for M=8. Every increase in Mach number by 1 cuts the power roughly in half, so a minimum of 20kW for M=6 is required. Therefore, it is recommended that an ablating model be made of camphor, if shape-changing studies are desired, and naphthalene, if manufacturing resources are limited. However, both materials are similar enough that the costs and benefits of choosing one or the other are entirely dependent on the situation and the preferences of the users/customers.

5.2 Operating Condition Recommendation

The use of a nozzle with a Mach number of 6 is recommended. This Mach number significantly reduces the heating power required by the heater while leaving a buffer to prevent sublimation at maximum heat flux. If the design of a new nozzle is not feasible, keeping the existing nozzle will still provide adequate testing of hypersonic applications, even if power limits the test section stagnation temperature to a point that will barely allow camphor or naphthalene to sublimate. The model is recommended to stay in the test section as no additional temperature is required and the process of installing new mounting mechanisms downstream would be difficult. In-line heaters and condensers can be found from various distributors.

5.3 Next Steps and Future Work

There are multiple ways to improve the analysis for this project. Assumptions were made when building the thermodynamic model to assume a constant reservoir temperature and to ignore boundary layer effects on the effective test section geometry. To improve the accuracy of the model, a quasi-steady temperature assumption could be made for the tank. As the air temperature within the tank decreases, more power is supplied from the heater to compensate for the lower stagnation temperature. Tran-
sient effects may be modeled as well, but the complexity of calculating these small transient effects may not make doing so ideal. Therefore, their affect on the heater power may be accounted for within the factor of safety instead. Additionally, the boundary layer thickness equation can be applied to alter the nozzle area ratio and effective size of the test section. Conducting another CFD analysis may improve the validity of the results. Using a coarser mesh in order to allow the model to converge at a larger time-step may improve the smoothness of the results and reduce the size of the scaled residuals. It may also be valuable to increase the complexity of the CFD simulation to capture some of the missing assumptions. One option is to capture the three-dimensional geometry of the tunnel by switching from an axisymmetric mesh to a three-dimensional one to capture the effect of corners, rectangular cross-sections, and pipe bends. Another option would be to add a turbulent model to the solver, which would help capture the effect of boundary-layer growth, shock interactions, and unsteady flow within the tunnel. The analysis may be able to be further validated by conducting an experimental test with the current tunnel. Upon finding a way to install thermocouples and other flow measurement devices into the tunnel, it may be possible to collect enough data to compare to the analytical and CFD results. Using the nozzle size recommendations, it would be valuable to utilize the method of characteristics to design a new isentropic nozzle to install into the tunnel. It would also be valuable to contact various wind-tunnel heater suppliers and receive estimates on the cost and installation process of a heater to the current facility.

Upon development of this project, there are many other projects that could be undertaken to elevate the possibilities and hypersonic regime captured by this facility. Adding a vacuum pump or vacuum chamber to the tunnel exit would allow for a higher pressure ratio, and therefore smaller nozzle and higher Mach number. Additionally, adding an electrode array to the test section could allow for forced ionization of the flow for analysis of chemical non-equilibrium effects.
BIBLIOGRAPHY


APPENDICES

Appendix A

ANSYS ICEM REPLAY SCRIPT

The model was constructed in Ansys ICEM CFD using a replay control script that allows small changes to be made to the parameters and mesh of a model without significantly changing how the model is generated. An example of this script is given in figure A.1.

Figure A.1: Replay script that controls model parameters.

The approximate lengths and cross-sectional hydraulic radius measured directly on the current tunnel were applied to the software by first defining points that encapsulate the size and shape of the tank, pipes, plenum chamber, test section, and diffuser to look similar to figure 2.1. Since the simulation is one-dimensional and axisymmetric, everything below the tunnel centerline was not constructed. Curves defining the walls, centerline axis, and important interior boundaries for building the structural
mesh were added connecting these points and surfaces defining the fluid were constructed out of the resulting geometries enclosed by the curves. The final geometry as seen in ICEM is shown in figure A.2.

Figure A.2: Replay script that controls model parameters.