Rocket Propulsion Design

Govind Chari

March 25, 2020

Foreword

This document is comprised of the information I learned in my first year on the Cornell Rocketry Team as a member of the Propulsion Subteam. During this time, I learned a great deal about combustion modeling and solid motor theory from by Daniel Morera, Matthew Schneider, and nakka-rocketry.net. I learned nozzle theory and design from online resources such as nakka-rocketry.net and aerodynamics4student.com and from the textbook *Elements of Rocket Propulsion* by Sutton.

This is meant as an introductory text/reference guide for solid propulsion and nozzle design. It is, in my opinion, a good starting place for those interested in propulsion. If this text gets math heavy at some points and seems intimidating, I would suggest that you focus your attention on the intuitive ideas behind the math rather than cramming equations into your head. I am a huge proponent of having an intuitive, first-principles understanding rather than focusing too much on the math and memorizing equations, so I try my best to include an intuitive explanation of anything I cover.

After reading though this text, if you are interested in this field, I would suggest designing a mock solid motor and nozzle yourself by creating the Matlab scripts described in this text given some thrust and impulse. The best test of knowledge is actually doing. If you want to learn more about propulsion, I would suggest reading *Elements of Rocket Propulsion*, as that text discusses propulsion much more in-depth than this text.

Contents

Chapter 1

Introduction

This document is divided into four parts: Solid Motor Theory, Combustion Modeling, Nozzle Theory, and Nozzle Design. At the end of reading this document, if you read carefully, you will understand how a rocket motor works thoroughly and be able to derive design parameters, such as chamber pressure, grain geometry, and nozzle expansion ratio from system level requirements like desired impulse and thrust. In addition, you will also be able to design the optimal nozzle based on these design parameters. Throughout this text, I strive to give the reader an intuitive understanding of the physical mechanisms governing the design, but also try to present enough mathematics, so this process can be replicated and fully understood quantitatively.

The first section, Solid Motor Theory, explores what a motor is, what it consists of, how it works, and discusses the parameters necessary to properly design a solid motor. The second section, Combustion Modeling, introduces a method that is used to derive design parameters from desired impulse and thrust. The third section, Nozzle Theory covers the basics of compressible fluid mechanics and dives into the math and physics which govern flow through a nozzle. The final section, Nozzle Design, covers how to design the most efficient nozzle contour using the method of characteristics and Rao's method.

To fully understand all of the information in this text, it is assumed that the reader is very comfortable with mechanics at the level of an introductory physics class, as well as a basic understanding of thermodynamics and fluid mechanics. In order to implement Combustion Modeling and Nozzle Design, the reader must know a coding language, ideally Matlab (unless you want to evaluate the Prandtl-Meyer Relation by hand). A good reader should take time reading and not rush through. Most of all, have perseverance. You will likely get frustrated at certain points because material isn't clicking fast enough in your head. After all, you are learning Rocket Engineering, but persist. Given enough mental effort, you can fully understand every sentence in this book.

Chapter 2

Solid Motor Theory

2.1 Motor Basics

A rocket motor is the propulsion unit of the rocket. It consists of the motor casing (combustion chamber), the grains (propellant), the igniter, and the nozzle. The motor casing is a long cylindrical tube that contains the grains inside of it, the igniter on one end, and the nozzle on the other end. In the image below, the "Metal Structure" is the motor casing.
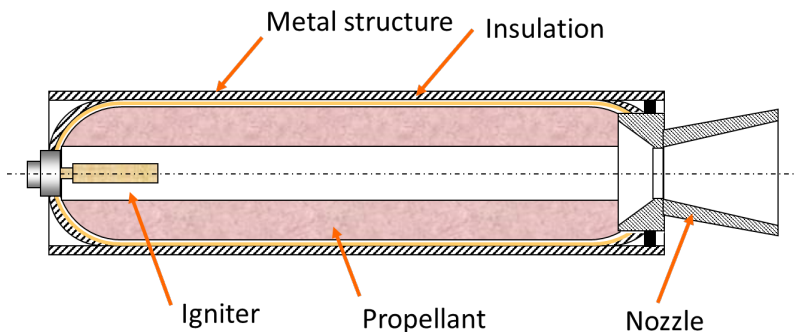
Figure 2.1: Motor Cross Section from www.narom.no/

A motor generates thrust from the conservation of momentum. The motor gives the hot gases it generates a significant amount of momentum in one direction, so the motor (and the rocket it is attached to) is propelled in the opposite direction.

The propellant is composed of a fuel and an oxidizer. When the propellant is ignited, the oxidizer generates a significant amount of oxygen and allows the fuel to burn. This reaction is very exothermic and generates many, many moles of combustion product (mainly gases). Since the gases are in a fixed-volume container (the motor casing) they are also very high pressure. As a result, these gases contain a significant amount of energy due to its hot temperature and high pressure. These hot, pressurized gases are then accelerated through the nozzle to the highest possible speeds (to get the most thrust) and thus, this thermal and pressure energy is converted to kinetic energy.

2.2   Compressible Fluid Dynamics Basics

In this section we will start to learn the basic physics needed to move on with the study of solid rocket motors. A more in-depth section on Compressible Fluid Dynamics will be presented in the Nozzle Theory section, however, it is important to have a basic understanding of this field before moving on with Solid Motor Theory. This section will present formulas and describe the intuition behind each of them, but will not present their derivations. The derivations can be found in any fluid dynamics textbook, or at http://www.aerodynamics4students.com/gas-dynamics-and-supersonic-flow/.

2.2.1   Compressible Flows

Fluid Dynamics can be be broadly separated into two groups: incompressible, and compressible. An incompressible fluid is one whose volume doesn't change significantly under pressure. This means that the density remains constant. Think of water. If you squeeze water, its volume doesn't really change. A compressible fluid is one whose volume does change when pressure is applied and consequently so does its density. Think of air. If you squeeze air, its volume shrinks, this is the whole principle behind air compressors.
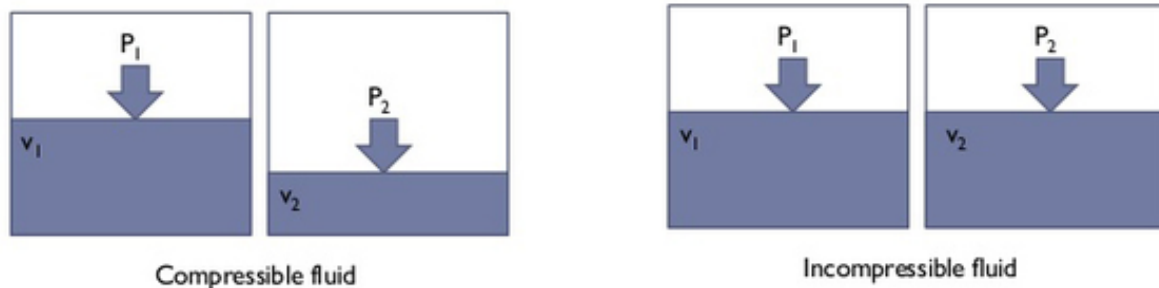
Figure 2.2:   Compressible vs Incompressible Fluids from https://www.quora.com/

When studying moving fluids, we can even treat air as incompressible if it isn't moving too fast. If air is moving slowly and hits a surface, its density doesn't really change, but if air is moving very fast and hits a surface, it gets compressed. But where do we draw the line? The rule of thumb is to treat airflow as incompressible if it is travelling slower than Mach 0.3 and treat it as compressible if it is travelling faster than Mach 0.3. Mach numbers of a flow are defined as the ratio of the speed of that flow to the speed of sound in that fluid. So if the combustion product of a motor is travelling twice the local speed of sound, we say it is travelling at Mach 2.

The exhaust gases of a rocket are well in excess of Mach 0.3, which makes compressible fluid dynamics a very useful tool to understand and design rocket propulsion systems.

2.2.2   Supersonic Flow

In addition to being compressible, most of the flow studied in propulsion are also supersonic, or are faster than the local speed of sound. Supersonic flows can behave strangely and lead to some counter-intuitive results. In order to understand these flows, it is important to understand what "local speed of sound" means.

The local speed of sound is the speed at which pressure waves propagate through the fluid. Pressure waves are how the fluid "communicates" changes in fluid properties.

The two equations below define Mach number, and give a formula to calculate the speed of sound in a fluid.

$$M = \frac{v}{a} \tag{2.1}$$

$$a = \sqrt{\gamma R T} \tag{2.2}$$

Equation 2.2 says that the speed of sound is proportional to the temperature of the gas. The speed of sound is the speed at which pressure waves can be propagated. It is limited by the speed of the gas molecules themselves. If you increase the temperature of the gas, then the gas molecules speed up and they can propagate pressure waves faster and thus the speed of sound in the gas increases. Similarly, if you increase the specific gas constant, which can be done by choosing a lighter gas, the speed of sound increases. If the mass of the gas molecules decrease, they move faster given the same kinetic energy, thus the speed of sound increases.

We said that pressure waves are how changes in fluid properties are communicated, however, this idea is abstract, so let's consider an example.

Imagine an object being placed in a subsonic airflow (the red lines in Figure 2.3). Even before contacting the object, the streamlines separate and start to go around the object, this is an experimental fact. Some air hits the object so the object becomes a source of pressure signals (the blue lines in Figure 2.3) that "warns" the air upstream that there is on object in the way. Having received this "warning" the air upstream begins to separate even before contacting the object.

Now let's analyze the same situation if the object were placed in a supersonic airflow. In this case again, some air hits the object and the object becomes a source of pressure signals. However, the pressure signals are travelling slower than the incoming flow, because the flow is moving faster than the local speed of sound and the pressure signals propagate at the local speed of sound. This results in the pressure signals bunching up a certain distance in front of the object. The airflow travels as if there is no object in the way, since it doesn't receive a "warning" until it reaches this bunching of signals. Once it does reach this point, the flow suddenly turns to avoid the object instead of the smooth turning that occurs in the subsonic case.

Understanding this example will be critical for understanding other properties of supersonic flows.
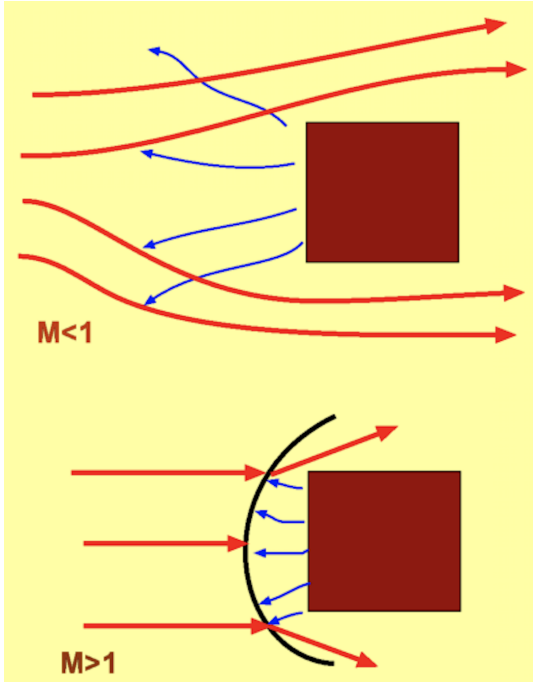
Figure 2.3: Subsonic vs Supersonic Flow from http://www.aerodynamics4students.com

2.2.3 Isentropic Flow Basics

As isentropic flow is by definition adiabatic and reversible. This means that no heat is added or removed from the flow, and the flow is free from any dissipative effects such as friction. The assumption of isentropic flow is the main assumption that we make when analyzing nozzles. This approximation is very good unless there is a shock-wave present. We will consider shocks much more in the Nozzle Theory section, but for now assume that all flow we deal with will be isentropic.

Two important thermodynamic constants we need to know are C_p and C_v. Both are properties of a gas, and under the assumption of a calorically perfect gas, these two quantities are constants and independent of pressure and temperature. The quantity C_p is the specific heat of of a gas that is under a constant pressure, and C_v is the specific heat of a gas that has constant volume. If the gas is heated at constant pressure, then as heat is added, the gas expands and does work on its surroundings. As a result, the heat you put into the gas goes into expanding it and raising its temperature. However, if you heat a gas at a constant volume, all of the heat goes into raising its temperature. From this is can be intuitively seen that C_p will always be greater than C_v. In addition, it can be proven that

$$C_p + C_v = R \tag{2.3}$$

Where R is the specific gas constant which is defined as the universal gas constant divided by the molar mass of that gas.

$$R = \frac{\overline{R}}{M} \tag{2.4}$$

8

From here on out, R will refer to the specific gas constant and $\overline{R}$ will refer to the universal gas constant. Furthermore, we can define another quantity called the ratio of specific heats which is defined as

$$\gamma = \frac{C_p}{C_v} \tag{2.5}$$

It can be seen that γ will always be greater than 1. This mysterious quantity arises in many, many texts, but is rarely presented with an intuitive explanation.

If a gas has a large C_p with respect to its C_v, you know it takes much more energy to raise the temperature of the gas at a fixed pressure than at a fixed volume. This means that the gas expands readily. Thus, gases with a higher ratio of specific heats expand readily and do more work than gases with lower ratio of specific heats. Phrased differently, if you have two gases with different ratio of specific heats in identical pistons, and if you were to add the same amount of heat to both pistons, the piston containing the gas with the larger ratio of specific heat does more work.

Another important equation is the ideal gas law, which is

$$p = \rho R T \tag{2.6}$$

Where p is the pressure of the gas, ρ is the density of the gas, R is the specific gas constant, and T is the temperature of the gas.

2.2.4 Isentropic Flow Relations

The pressure exerted by a moving fluid can be broken into two parts: static pressure and dynamic pressure. Static pressure is the pressure that you would measure if you measure the pressure that a flow exerts, and dynamic pressure is the kinetic energy per unit volume of a flow. The sum of these two parts yields the stagnation (or total) pressure of the flow. This makes sense coming when considering Bernoulli's Principle. If a fluid's velocity increases, its static pressure decreases. The stagnation pressure of a flow is the pressure that the gas would exert if brought to rest isentropically. In an isentropic flow, stagnation pressure is constant at all points in the flow, however the static and dynamic pressures can change at each point. Mathematically, this corresponds to the formula

$$p_0 = p_{static} + \frac{\rho v^2}{2} \tag{2.7}$$

where p_0 is the stagnation pressure, ρ is the density of the fluid in the flow, and v is the speed of the flow.

Temperature has a similar property. The stagnation temperature of a flow is the temperature that the flow will have if it is brought to rest isentropically. This can be seen when considering a plane in a supersonic wind-tunnel. As the air is flowing, its temperature isn't too high, yet when it is brought to rest by coming into contact with the plane, its temperature increases greatly. This extra thermal energy doesn't just appear all of a sudden. It is contained in the kinetic energy of the gas. When the gas is brought to rest, the kinetic energy is converted to thermal energy, and thus the temperature of the plane's exterior increases. Mathematically, this can be represented by

$$T_0 = T + \frac{v^2}{2C_p} \tag{2.8}$$

where T_0 is the stagnation temperature, T is the local temperature at any point in the flow, and v is the speed of the flow at the point corresponding to T.

Since the static pressure and temperature vary locally with velocity of the flow, density of the fluid must also vary with flow velocity from the ideal gas law. However, the relationship isn't as straight forward as it was for pressure and temperature above. This is because the density of a flow is related to BOTH the pressure and the temperature. From the equations 2.6, 2.7 and the ideal gas law, we can come up with some relations that relate the stagnation pressure, temperture, and density to the local pressure, temperature, and density in terms of the local Mach number. These relations are known as the isentropic stagnation relations. In addition there are a few more important definitions and relations that all together constitute the Isentropic Flow Relations.

$$\left(\frac{p_1}{p_2}\right) = \left(\frac{\rho_1}{\rho_2}\right)^\gamma = \left(\frac{T_1}{T_2}\right)^{\frac{\gamma}{\gamma-1}} \tag{2.9}$$

$$\frac{p_0}{p} = \left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{\gamma}{\gamma-1}} \tag{2.10}$$

$$\frac{T_0}{T} = \left(1 + \frac{\gamma-1}{2}M^2\right) \tag{2.11}$$

$$\frac{\rho_0}{\rho} = \left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{1}{\gamma-1}} \tag{2.12}$$

$$\frac{A}{A^*} = \left(\frac{\gamma+1}{2}\right)^{\frac{-\gamma+1}{2\gamma-2}} \frac{\left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{\gamma+1}{2\gamma-2}}}{M} \tag{2.13}$$

Equation 2.9 relates the pressure, density, and temperature between two points in an isentropic flow, and Equations 2.10-2.12 are the stagnation relations. The stagnation relations are used to express the pressure, temperature, and density in terms of the stagnation properties, which are much easier to find. You just need to find a stationary part of the flow, usually a reservoir upstream and measure its pressure and temperature there.

Equations 2.10-2.12 just say that if you are looking at a fast moving part of the flow, the pressure, temperature and density are lower than the same properties at their stagnation point or at any point where the flow is moving slower.

Equation 2.13 allows you to determine the Mach number at any portion of a duct if you know the cross-sectional area of the duct at that point and the cross sectional area at the throat. Equation 2.13 is derived from the continuity equation and equations 2.9-2.12. This equation will be explored more in the nozzle theory section.

2.2.5 Choked Flow

In a previous section, I mentioned that supersonic flow leads to many counter-intuitive properties. One such property is the idea of choked flow. This idea has a lot to do with how a nozzle functions, but knowing the basics will be important to understand Solid Motor Theory. This idea will be explored more in greater detail in the section on Nozzle Theory.

Normally, if a flow is constricted, it speeds up in order to keep mass flow rate constant. This is the Venturi Effect. However, choked flow is the point where this relationship breaks down. If you keep constricting a flow, the flow will only speed up to the local speed of sound, you cannot make the flow faster than Mach 1 by further constricting it. In addition, the mass flow rate cannot be increased by decreasing the pressure downstream. This is a rather abstract definition, so we will consider an example.

Consider flow through a pipe as shown below. The fluid moves from section one though a constriction into section two (upstream pressure is greater than downstream pressure). From the Venturi Effect, we know that the fluid will be faster in section two than in section one. By conservation of mass, we know that the mass flow rate through section one is the same as the mass flow rate through section two.

Now imagine we lower the downstream pressure. This will cause the flow in both sections to speed up, and thus the mass flow rate through the pipe increases. This is because the pressure differential between the two ends of the tube is increased. Now if we keep lowering the downstream pressure, we will reach the point where the flow in section two reaches Mach 1. Now, if we continue lowering the downstream pressure, this information cannot be propagated upstream, since this pressure information is propagated through the fluid at the speed of sound. Thus, decreasing the downstream pressure will not result in an increase in mass flow rate or fluid velocity. When this point is reached, we say we have choked flow.

One important thing to keep in mind is that while decreasing the downstream pressure will not result in an increase in mass flow rate when flow is choked, an increase in upstream pressure will increase the mass flow rate through the system. This is because increasing the upstream pressure will increase the density of the fluid, and thus increases the mass flow rate.
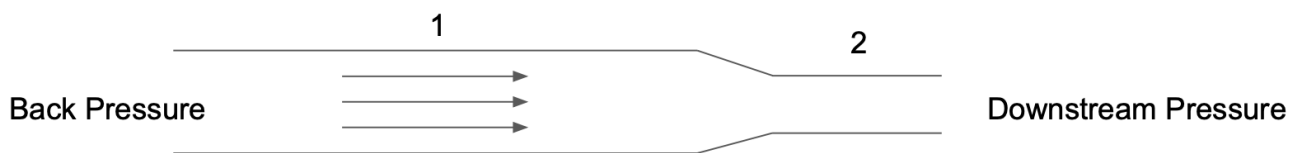

Figure 2.4: Choked Flow

Choked flow allows for a convenient way to calculate the mass flow rate at any point in the system. The formula for mass flow rate is

$$\dot{m} = \rho v A \tag{2.14}$$

Where $\dot{m}$ is mass flow rate at some point, v is the velocity of the flow at that point, and A is the cross-sectional area of the pipe at that same point.

We know that the mass flow rate at all points of the system is the same due to conservation of mass. We know the cross sectional area of the flow at its minimum point (A^*). We also know that its velocity is Mach 1 since flow is choked, so we can use Equation 2.2 to calculate this velocity. We can apply isentropic flow relations to calculate the density of the flow at the point of minimum cross-sectional area. In the end, we are left with an expression for the mass flow rate of a choked system in terms of stagnation pressure and temperature, which are usually known.

$$\dot{m} = \frac{p_0 A^*}{\sqrt{T_0}} \sqrt{\left[\frac{\gamma}{R} \left(\frac{2}{\gamma + 1} \right)^{\frac{\gamma+1}{\gamma-1}} \right]} \tag{2.15}$$

where A^* is the minimum cross-sectional area of the pipe, also known as the throat.

If we increase the back pressure of the gas, we can get a larger mass flow rate, since the density of the gas increases. If we increase the temperature of the gas, its density decreases, so our mass flow rate decreases. If we increase the minimum cross-sectional area, A^*, and flow remains choked, it makes sense that our mass flow rate will increase, since the opening for gas to exit through increases in size.

There are two important takeaways from this. When you have choked flow, the velocity of the gas at the minimum area is Mach 1 and the mass flow rate is fixed throughout the system and easy to calculate via Equation 2.15.

This concludes the section on the Compressible Fluid Dynamics. Now we will start to look at how this can be applied in designing a motor for a given thrust and impulse.

2.3   A First Look at Thrust

Since thrust is generated due to conservation of momentum, we can derive an expression for thrust by differentiating the momentum of the gas.

$$F_{thrust} = \dot{m} v_e + (p_e - p_a) A_e \tag{2.16}$$

Where v_e is the exit velocity of the gas out of the nozzle, p_e is the exit pressure of the gas, p_a is the ambient atmospheric pressure, and A_e is the exit area of the nozzle. The first term in the equation comes from the conservation of momentum, and the second term accounts for thrust generated due to a pressure difference between the combustion gases and the atmosphere. This second term, or pressure thrust, is usually not that significant when compared to the the first term.

When trying to maximize thrust, we can increase either the mass flow rate, or the velocity of the gas. In this chapter, we will focus on the former, and in the chapter on nozzles, we will discuss how to maximize the latter

If we assume that flow is choked (which it is most of the time), we can substitute Equation 2.15 into Equation 2.16. We can also use isentropic flow relations to get an expression for v_e in terms of stagnation pressure and temperature. We will analyze how to get the equation for v_e in the nozzle section, but for now just know that we can get it. We are then left with the following equation for thrust

$$F_{thrust} = p_0 A^* \sqrt{\frac{2\gamma^2}{\gamma-1}\left(\frac{2}{\gamma+1}\right)^{\frac{\gamma+1}{\gamma-1}}\left[1-\left(\frac{p_e}{p_0}\right)^{\frac{\gamma-1}{\gamma}}\right]} + (p_e - p_a)A_e \qquad (2.17)$$

This can be written in a more compact form that provides more intuition.

$$F_{thrust} = p_0 A^* C_f \qquad (2.18)$$

Where C_f is the thrust coefficient and is defined as

$$C_f = \sqrt{\frac{2\gamma^2}{\gamma-1}\left(\frac{2}{\gamma+1}\right)^{\frac{\gamma+1}{\gamma-1}}\left[1-\left(\frac{p_e}{p_0}\right)^{\frac{\gamma-1}{\gamma}}\right]} + \frac{(p_e - p_a)A_e}{p_0 A^*} \qquad (2.19)$$

In both of these equations p_0 is the chamber pressure which is also the stagnation pressure. This is because the velocity of the gas in the chamber is essentially 0 and insignificant when compared to the velocity of the gas in the nozzle. The quantity A^* is the cross sectional area of the throat of the nozzle, and p_a is the ambient pressure.

As can be seen from Equation 2.18, the main way we can increase thrust is by increasing chamber pressure. Although we could also increase the throat area as well, this decreases the chamber pressure so it won't affect thrust much. This makes sense, since if there is a smaller hole for the gas to escape, the pressure will be higher in the motor casing. We will investigate this relationship much more in the next section.

When we fire a motor, we can plot the thrust it generates over time. If we integrate this curve, we are left with the total impulse of the motor. So if we want to increase the impulse of a motor, we can either make it fire for a longer period of time, or we can increase the thrust it produces. Both factors can be controlled by considering the grain geometry, which we will look at next.

2.4 Grain Geometry

2.4.1 Grain Basics

A grain is a unit of solid fuel. It is a cylinder of fuel with a hole through the center. This hole allows the combustion product to flow through the center of the motor and out through the nozzle. A motor usually has multiple smaller grains than one long grain. The image below shows multiple grains separated by spacer rings.

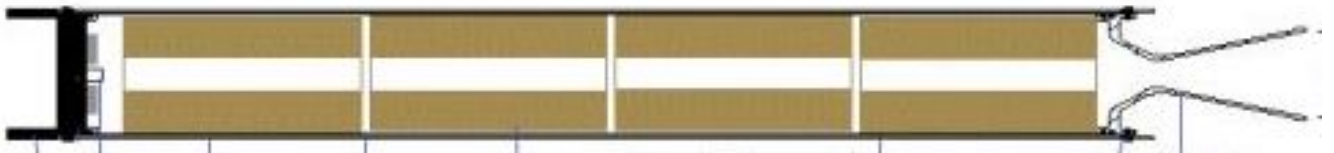**Figure 2.5:**   Motor With Multiple Grains from https://space.stackexchange.com/

Each grain does not have to have a constant diameter circular bore through the center. There are many other shapes that can be chosen. Each shape will result in different pressure

vs time curves as the grains burns. In the image below, many different shaped cores are shown with their respective thrust vs time curve.
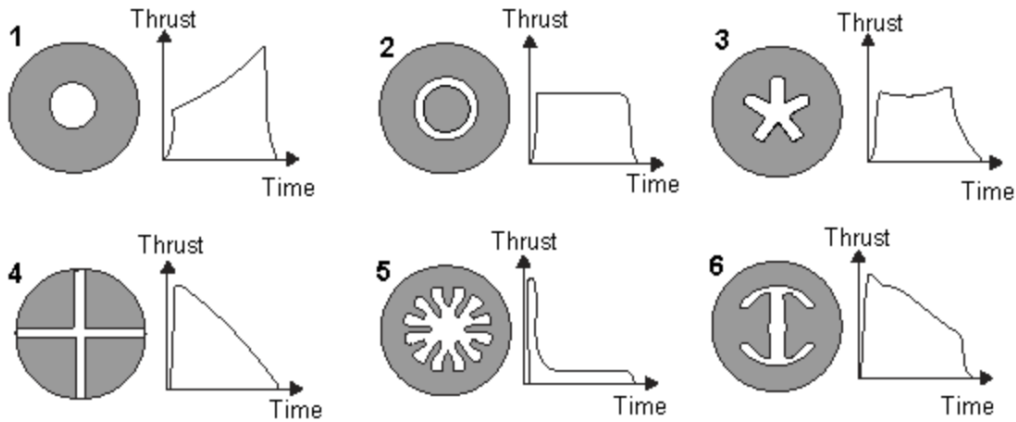

Figure 2.6: Core Geometries and Thrust Curves from https://space.stackexchange.com

This text will focus on the BATES grains (grain 1 in Figure 2.6) for simplicity, but the same ideas apply to other grain shapes, but their analysis is more complicated. When designing a motor, the main thing to consider is grain geometry. This refers to the number of grains you will use, their respective length, their outer diameter, and their core diameters. The outer diameter of the grain is usually already fixed. The outer diameter of the grain is restricted by the inner diameter of the motor casing, whose outer diameter is restricted by the diameter of your rocket's airframe. Since, this factor cannot be chosen, you are left with deciding the number of grains you want, their lengths, and their core diameters.

When designing a motor you should always keep in mind the desired thrust and impulse. These are the design specifications that the motor must meet. As mentioned in the previous section, both of these are influenced by the chamber pressure vs time curve, which in turn is influenced by the grain geometry. I have said this a number of times, so now let's jump right into it.

2.4.2 Grain Burning Physics

The burning of grains is a highly exothermic reaction that generates many moles of gas. The buildup of this gas in the motor casing is what causes chamber pressure.

Below, a BATES grain is shown. Its two ends and the inner surface (port) are the surfaces that burn. The outer surface is inhibited (doesn't burn), since it is bonded to the inside of the motor casing. In the image, the x with the yellow arrow shows how the dimensions of the grain change in one time step. The diameter increases by $2x$ and the length decreases by $2x$.
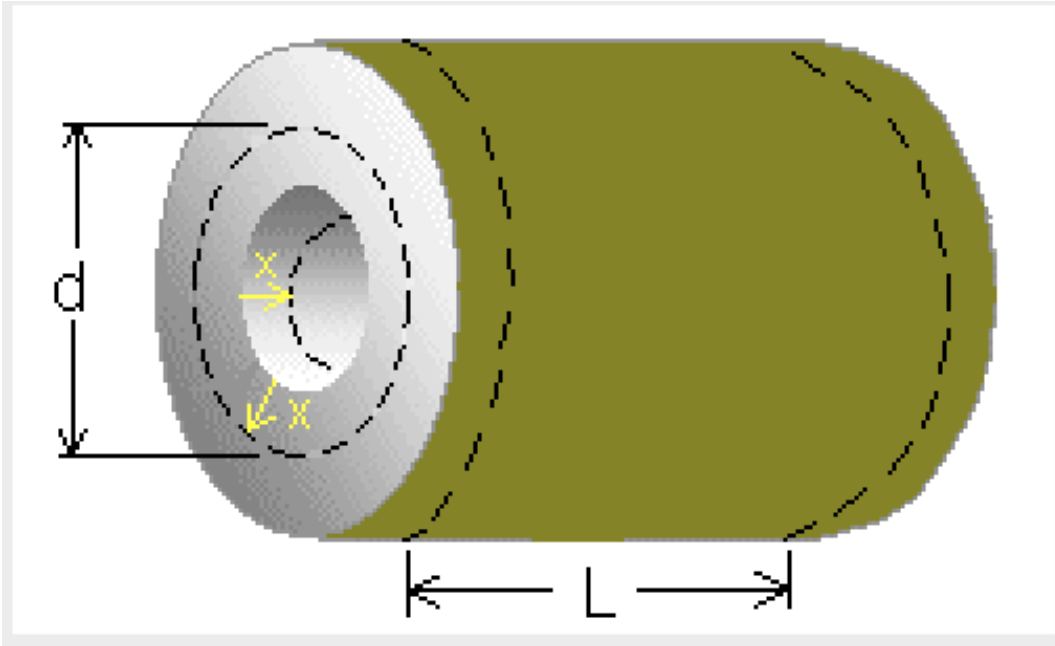
Figure 2.7: BATES Grain Surface Regression from http://www.nakka-rocketry.net

The burn rate is defined as the regression distance x divided by the length of the time step. The burn rate increases as the chamber pressure increases. The relationship between burn rate and chamber pressure is given by Saint Robert's Law.

$$r = ap_0^n \tag{2.20}$$

Where a and n are constants that must be derived empirically. These constants are unique to the fuel that is used. In addition, values of a and n are only valid in 300psi pressure ranges. So you will have a and n for the 0-300psi pressure range and a different a and n for the 300-600psi pressure range.

2.4.3 Understanding Pressure Curves

The image below shows a standard pressure curve that you would get from burning a BATES grain.
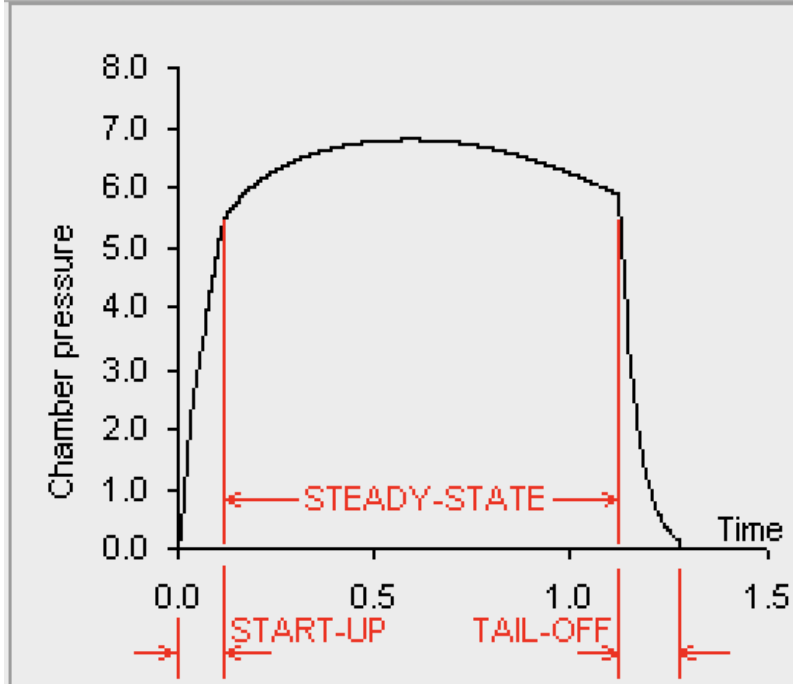
Figure 2.8: Pressure vs Time Curve from http://www.nakka-rocketry.net

As labeled in the image, there are three main phases of the burn: start-up, steady-state, and tail-off.

During the start-up phase, the grains start to burn and the chamber-pressure starts to build up. During this quick phase, chamber pressure greatly increases as choked conditions are established. During steady-state, the rate that gas that is being generated is equal to the rate at which the gas is leaving the combustion chamber. Since choked conditions are established, this mass generation rate can be calculated through Equation 2.15. In this phase the pressure doesn't vary wildly, and the only variation in pressure is due to changes in the grain geometry. The steady-state phase is what is tried to control though grain geometry, as this phase makes up the vast majority of the burn time and thus generates almost all of the impulse. The tail-off phase is when the grains have finished burning and the motor casing depressurizes. This phase along with start-up generate negligible impulse.

Using a mass balance, applying isentropic flow relations and Saint Robert's Law, a differential equation for chamber pressure can be derived. (There is an amazing derivation of this on www.nakka-rocketry.net on the chamber pressure webpage)

$$\frac{V_0}{RT_0}\frac{dp_0}{dt} = A_b a p_0^n (\rho_p - \rho_0) - p_0 A^* \sqrt{\frac{\gamma}{RT_0} \left(\frac{2}{\gamma+1}\right)^{\frac{\gamma+1}{2\gamma-2}}} \tag{2.21}$$

Where V_0 is the free volume in the combustion chamber, A_b is the surface area of the burning surface, A^* is the throat area of the nozzle, and ρ_p is the density of the propellant.

This differential equation can be solved to find the pressure during start-up. During steady state $\frac{dp_0}{dt} = 0$, so Equation 2.21 can be reduced to

$$p_0 = K_n \rho_p r c^* = (K_n \rho_p a c^*)^{\frac{1}{1-n}} \tag{2.22}$$

16

Where K_n is the dimensionless Klemmung Number and c^* is the characteristic velocity, which is a measure of combustion efficiency which we will explore more in the Nozzle Theory Chapter. They are defined as

$$K_n = \frac{A_b}{A^*} \tag{2.23}$$

$$c^* = \sqrt{\frac{RT_0}{\gamma \left(\frac{2}{\gamma+1}\right)^{\frac{\gamma+1}{\gamma-1}}}} \tag{2.24}$$

The main relationship that should be understood in Equation 2.22 is that as the surface burn area increases, the chamber pressure increases, and as the throat area decreases, the chamber pressure increases. This relationship can be intuitively understood. If the burn area is larger then more gas is being generated, which would increase chamber pressure. If throat area were decreased, then the gas could escape slower from the motor casing and more gas will accumulate, so the pressure will be higher. However increasing pressure by decreasing throat area will not affect thrust much, since thrust is directly proportional to throat area by Equation 2.18.

Thus, the main way to control the pressure curve in a way to control thrust is by changing the burn area of the grain.

2.4.4   Manipulating Pressure Curves

For a BATES grain, there are two factors at play when analyzing the burn area of the grain over time. Since the grain burns from inside out, the inner circumference increases as the grain burns. However, the length of the grain decreases as the grain burns as well. We can come up with an equation for burn area as a function of the distance burned through, using geometry.

$$A_b = \pi \left(d_0 + 2x\right)\left(L_0 - 2x\right) + 0.5\pi \left(D^2 - \left(d_0 + 2x\right)^2\right) \tag{2.25}$$

Where d_0 is the initial port diameter, L_0 is the initial grain length, D is the outer diameter of the grain, and x is the distance burned through.

Since the burn rate is more or less constant during the steady-state phase of the burn, the burn area vs time curve has the same shape as the burn area vs distance burned curve. We can see, that if L_0 is large, then the inner circumference increase of the grain is more significant that the length decrease of the grain. As a result, for long grains, the burn area tends to increase with time. On the flip side, for shorter grains, the length decrease is more significant than the inner circumference increase, so for shorter grains, the burn area tends to decrease with time.

If burn area (and thus pressure) tends to increase with time, the pressure curve is said to be progressive, and if the pressure tends to decrease with time, the pressure curve is said to be regressive. If the pressure tends to stay constant over time, the pressure curve is neutral.

Referring to Figure 2.6, we can see that thrust curves 2 and 3 are neutral, 1 is progressive, and 4, 5 and 6 are regressive.

If you want to increase the thrust of your motor, then you need to increase the burn area of the grains. You can do this by using multiple grains or by increasing the length of each grain. If you want to increase the delivered impulse of your motor, you either need to increase the thrust or the burn time of the motor. You can increase the burn time of the motor by increasing the inner diameter of the grain as the grain tends to burn to completion due to increases of the inner circumference rather than decreases in the length of the grain. By decreasing the inner diameter of the grains, you allow the grain to burn for a longer time before it is consumed. However, decreasing the grain diameter too much can lead to erosive burning concerns.

2.4.5 Erosive Burning

One very important factor that needs to be considered when selecting a grain geometry is erosive burning. Erosive burning is an acceleration of the grain burn rate due to gas moving parallel to the burn surface. If you have alcohol on a table and you blow parallel to the table surface, the evaporation rate of the alcohol increases. This is a similar principle to erosive burning.

This increase in burn rate leads to an increase in chamber pressure by Equation 2.22 and thus an increase in thrust by Equation 2.18. Erosive burning can be a good thing as it results in a thrust increase, but it can also be detrimental.

If the mass flux (mass flow rate divided by port cross sectional area) gets too large, the fast moving gas can break chunks off of the grains it is flowing parallel to. If chunks are broken off, then the burning area suddenly increases, which leads to a sudden increase in pressure, which leads to an increase in burn rate, which increases the pressure further. This can cause the motor casing to explode.

2.5 Conclusion

In summary, the grain geometry (number of grains and dimensions of grains) is directly related to chamber pressure by Equation 2.22. The chamber pressure is related to thrust by Equation 2.18, and integrating thrust over time yields delivered impulse of the motor.

If you want to make the thrust curve more regressive, decrease the length of the grains or split longer grains into multiple shorter grains. If you want to make the thrust curve more progressive, increase the length of each grain or combine shorter grains into longer grains. If you want to increase the thrust of the motor, you should increase the burn area by increasing the number of grains used and/or increasing the length of the grain. If you want to increase the delivered impulse of the motor you can increase the thrust of the motor, or decrease the port area of the grain (which increases burn time), but make sure to consider erosive burning.

Chapter 3

Motor Design

Soon to come!

Chapter 4

Nozzle Theory

4.1   Nozzle Basics

After the grains ignite in the combustion chamber you have high pressure, high temperature gas that will be harnessed to produce thrust. You may be asking yourself, "Why can't you poke a hole in the bottom of the combustion chamber and let that gas escape and produce thrust? Why do you need a nozzle?"

The answer to this question is that you want to accelerate the gas to its fastest possible speed to produce the most thrust, and having this gas escape through a hole will not accomplish this goal. In order to accelerate this gas to its fastest speeds, we need a nozzle with a very specific geometry. In this chapter we will explore the physics behind how a nozzle works.

4.2   Nozzle Geometry

A rocket nozzle, converging-diverging nozzle, or deLaval nozzle can be divided into three parts: the converging section, the throat, and the diverging section. In the converging section, the gas is subsonic (below Mach 1), at the throat, the gas is sonic (remember the section on choked flow), and in the diverging section, the gas is supersonic.
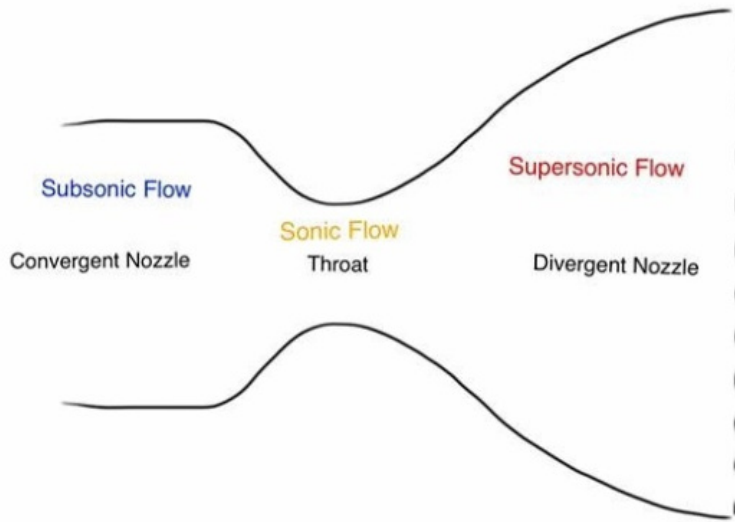
Figure 4.1: Nozzle Anatomy from hybridrocketengine.wordpress.com/

As the gas moves through the nozzle, thermal and pressure energy are converted to kinetic energy. Thus, pressure and temperature decrease while gas velocity increases. Calculating exact values of temperature, pressure, and velocity can be performed by utilizing isentropic flow relations.
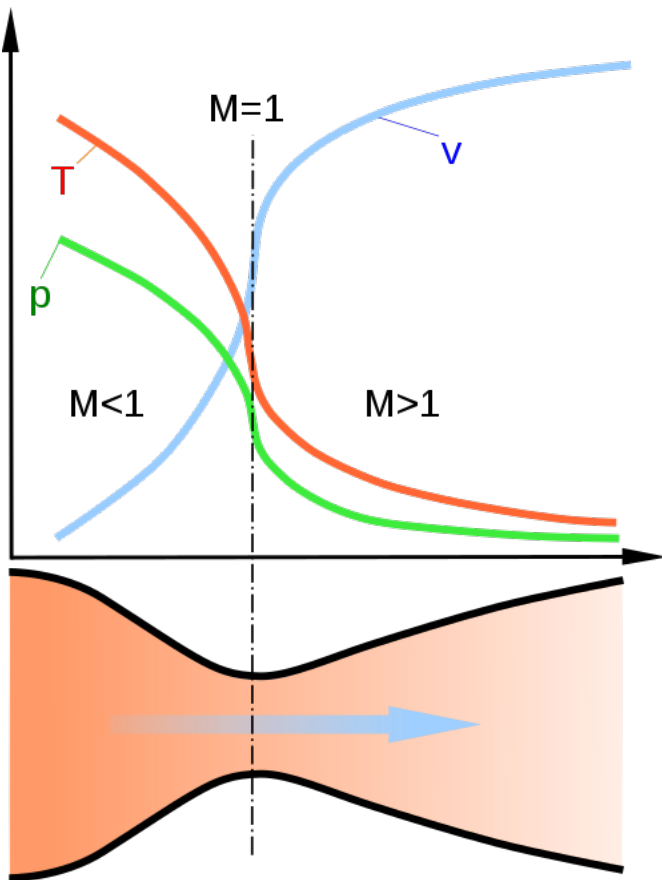
4.3 Expansion

4.3.1 Assumptions and Conservation Laws

Before going any further with the analysis of nozzle flows, we need to clearly state the assumptions made in this section, as well as the conservation laws that we will use.

For ideal nozzle analysis, we assume that the flow is isentropic, the gas is an ideal gas, the flow through the nozzle is steady (not time varying), the only combustion product is gas (there are no solids or liquids), and flow is 1D (we assume the flow is only exiting the nozzle axially). The isentropic assumption is a good one to make, as long as there are no shocks in the nozzle. The steady-state assumption is valid for the steady-state part of the burn as mentioned in Chapter 2. However, there will be solids in the combustion product, and we will discuss how that slightly changes our analysis later, but for know we will assume that the only product is gas.

The two major conservation laws that we will be using are conservation of mass (continuity) and conservation of energy. For a steady flow, which we assume, conservation of mass through a nozzle states that all parts of the nozzle has the same mass rate through it. Mathematically, this is

$$\rho_1 A_1 v_1 = \rho_2 A_2 v_1 \tag{4.1}$$

Where, 1 and 2 are any two arbitrary points in the nozzle, and ρ is density of the gas at that point, A is the cross-sectional area at that point, and v is the velocity of the gas at that point.

Conservation of energy states that the stagnation enthalpy of a flow remains constant. Enthalpy is the amount of energy available for heat transfer (either as heat or work), and stagnation enthalpy is the enthalpy of the flow if it were brought to rest.

This principle follows a similar definition to stagnation pressure and temperature. The enthalpy of a flow decreases as the flow velocity increases. This makes sense, since enthalpy is a measure of the total thermal energy that a flow contains. So if a flow moves faster, it gains kinetic energy which means it must be losing thermal energy, or enthalpy. Mathamatically, this is

$$h_0 = h + \frac{v^2}{2} \tag{4.2}$$

Where h_0 is the stagnation enthalpy, h is the enthalpy of a flow at any arbitrary point in the nozzle, and v is the velocity of the flow at that point.

If we apply the definition of specific heat

$$C_p = \frac{\Delta h}{\Delta T} \tag{4.3}$$

to Equation 4.2, we get Equation 2.8. Thus, equations 2.8 and 4.2 can both be thought of as conservation of energy. In essence, they both say that the energy of a flow can either be in the thermal energy of the flow, or the kinetic energy of the flow and that the total energy remains constant.

4.3.2   Area-Mach Relation

The way that a nozzle accelerates a gas is by varying its cross-sectional area. For subsonic flows you can accelerate the gases by decreasing the cross-sectional area of the nozzle. Consider blowing air out of your mouth. Open you mouth widely and blow air. Then purse your lips and blow again. In the second case, the air moves faster. This shows that if you have subsonic flow and you decrease the cross-sectional area of the nozzle, the flow speeds up.

However, you can only speed up a flow until Mach 1 by constricting it. This was touched upon in the section on choked flow. If you want to accelerate a sonic flow past Mach 1 into the supersonic regime, you need to expand the flow by increasing the cross-sectional area. This is why nozzle initially converge, then diverge. The gas is accelerated in the converging section until Mach 1, which occurs at the throat, then the diverging section accelerates the gas past Mach 1.
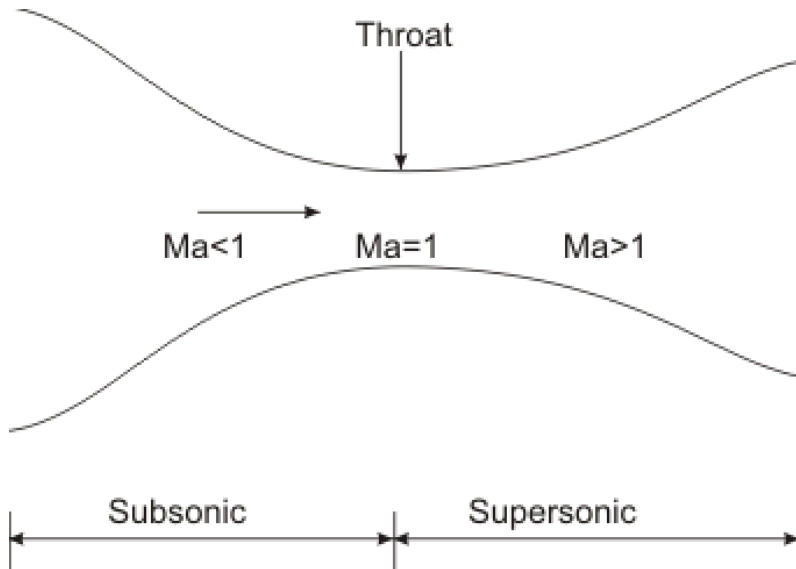

Figure 4.3:   Converging-Diverging Nozzle from nptel.ac.in/

This idea isn't intuitive at all, since we don't really have any daily experiences with supersonic flows. The reason that sonic flows need to be expanded to accelerate them further is the same reason that subsonic flows need to be constricted to accelerate them: the continuity equation.

From the continuity equation (Equation 4.1), if area is decreased, then some combination of increasing the density and the velocity must occur. When we are at low Mach numbers, the density doesn't change by much, so if you have subsonic flow the flow will speed up in order for mass flow rate to be constant. This explains the converging section.

23

The diverging section is a bit trickier. If you were to increase the area, then some combination of decreasing density and velocity must occur to keep the mass flow rate constant. However, these two quantities move in opposite directions. As flow speeds up, the front of a fluid element speeds away from the back, so the element will take up more space and thus its density decreases. The Mach number of the flow determines how strong this inverse correlation is. When flow is subsonic and a fluid element decelerates, the flow behind it smoothly decelerates, since the deceleration of the element in front can be communicated through pressure signals to the flow behind it. Since the deceleration is smooth, there is only a mild density increase of the fluid. However, if flow is supersonic and a fluid element decelerates, this information cannot be propagated upstream, since the flow is moving faster than pressure waves propagate. This results in the flow behind the element decelerating very quickly and compressing itself, which results in a sharp increase in density. Thus, if the Mach number is one or greater, decreasing the flow will sharply increase the density. However, if area is increased, the product of density and velocity must decrease, and the only way for this relationship to be satisfied is if the velocity increases and the density decreases sharply.

In short, when flow is subsonic, density changes can be ignored, so if flow is constricted, it must speed up. However, for supersonic flows, density changes are very large and in the opposite direction to velocity changes, so if the flow is expanded, the flow must speed up in order for continuity to be satisfied.

This can be quantitatively expressed in the Area-Mach Relationship

$$\frac{dA}{A} = (M^2 - 1)\frac{dv}{v} \tag{4.4}$$

Where dA represents a change in cross-sectional area, dv represents a change in velocity, and M is the Mach number of the flow.

It can be seen that if we have a subsonic flow, $M < 1$, and we decrease the cross-sectional area, $dA < 0$, then the flow speeds up, $dv > 0$.

If we have a supersonic flow $M > 1$, and we increase the cross-sectional area, $dA > 0$, then the flow speeds up, $dv > 0$.

Most interestingly, if we have a sonic flow, $M = 1$, we see that $dA = 0$. This means that a sonic flow can only exist at a local minimum of cross-sectional area. This proves that sonic flow can only exist at the throat of a nozzle, where cross-sectional area is at a minimum.

4.3.3   Expansion Ratio

Now, as promised, we will be taking a closer look at Equation 2.13. This equation is very important when calculating the Mach number of gases at the exit of the nozzle.

If we start with the continuity equation with one term representing the mass flow rate at some arbitrary point in the nozzle and the other term representing the mass flow rate at the throat, they are, of course, equal.

$$\rho v A = \rho^* v^* A^* \tag{4.5}$$

Where ρ^* is the density at the throat of the nozzle, v^* is the velocity of the gas at the throat of the nozzle, which is Mach 1, and A^* is the cross-sectional area of the throat. We

can then apply isentropic flow relations to all quantities but the two areas and we are left with the expression

$$\frac{A}{A^*} = \left(\frac{\gamma+1}{2}\right)^{\frac{-\gamma+1}{2\gamma-2}} \frac{\left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{\gamma+1}{2\gamma-2}}}{M} \tag{4.6}$$

With this isentropic relation and the others, if we know the stagnation pressure and temperature (the pressure and temperature in the combustion chamber), we can calculate the pressure, temperature, density, Mach number, and velocity of the flow at any point in the nozzle if we know the cross-sectional area at that location and if the nozzle doesn't have any shocks in it. I cannot overemphasize how powerful of a tool this is.

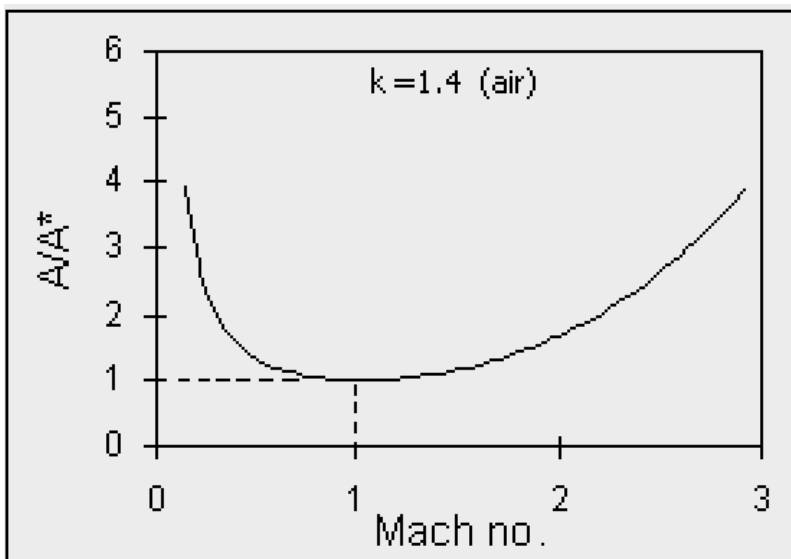We can also plot $\frac{A}{A^*}$ versus Mach number to gain some insight.

Figure 4.4: $\frac{A}{A^*}$ vs M from nakka-rocketry.net

As you can see on the graph, if you have flow under Mach 1, it can be sped up by decreasing the cross-sectional area of the nozzle. The only place where the flow is sonic is when $\frac{A}{A^*} = 1$, which is at the throat. Then once sonic flow is achieved, the only way it can be accelerated further is if the area is increased. We can also see that the larger the cross-sectional area becomes, the faster the flow is.

This leads us to define a new quantity, the expansion ratio

$$\epsilon = \frac{A_e}{A^*} \tag{4.7}$$

Where ϵ is the expansion ratio and A_e is the area of the nozzle at the exit.

From Equation 4.6, we can see that the larger the expansion ratio, the faster we can accelerate the gas to. Now this raises a question, if we can accelerate the flow to its highest possible speeds by increasing the exit area of the nozzle, why don't we just design a nozzle with a massive expansion ratio to yield the fastest gas and most thrust?

4.3.4   Underexpanded, Ideally Expanded, and Overexpanded Flows

From our isentropic flow relations, we know that the more we expand a flow, the faster it becomes (Equation 4.6), and the faster a flow becomes, the lower the pressure becomes (Equation 2.10). So, the more we expand a flow, the lower the pressure becomes. As the supersonic flow exits the nozzle, we have three possible conditions, underexpanded, ideally expanded, and overexpanded.

Underexpanded flows occur when the exit pressure of the gas is higher than the ambient pressure. This occurs if we don't expand the gas enough. When this occurs, the gas spreads out as it exits the nozzle, since the higher pressure exhaust pushes outwards against the lower pressure atmosphere. This is shown in the top image of Figure 4.5

Ideally expanded flows occur when the exit pressure of the gas is equal to the ambient pressure. This means that we expanded the gas enough. When this occurs, the gas exits straight out of the nozzle, since the gas and atmosphere are at the same pressure, so neither pushes against each other. This is shown in the second image of Figure 4.5.

Overexpanded flows occur when the exit pressure of the gas is less than the ambient pressure. This occurs if we expand the gas too much. When this occurs, as the gas exits the nozzle, it gets compressed inwards due to the higher pressure atmosphere. This is shown in the third image in Figure 4.5. If a flow gets expanded so much that it detaches from the nozzle wall, it is referred to as grossly overexpanded. This is shown in the last image in Figure 4.5.
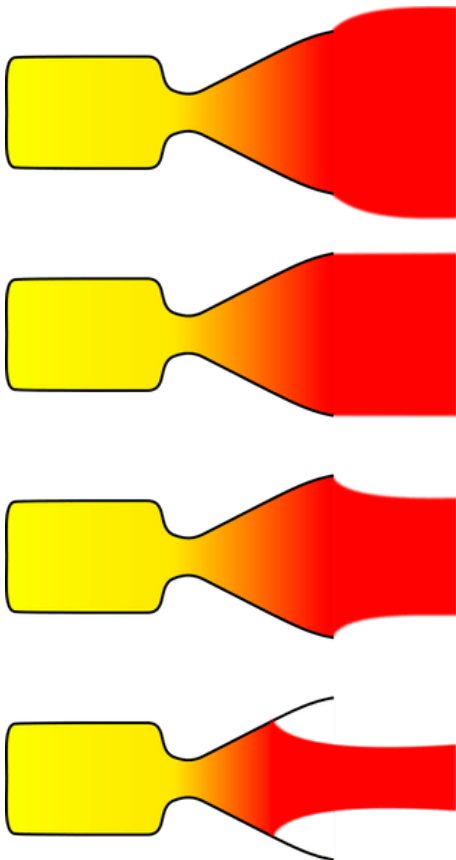

Figure 4.5:   Under, Ideally, and Overexpanded flows from www.wikipedia.org

In overexpanded flows, the exhaust gets adiabatically compressed and decelerated, so exhaust heats up (by Equation 2.11), and any unburned fuel gets reignited in a regular pattern called Mach diamonds. The mechanism behind why the pattern repeats is alternating oblique shocks and expansion fans, but don't worry about that right now. They look super cool. The image below shows Mach diamonds generated by the J58 engine (the engine of the SR-71 Blackbird).
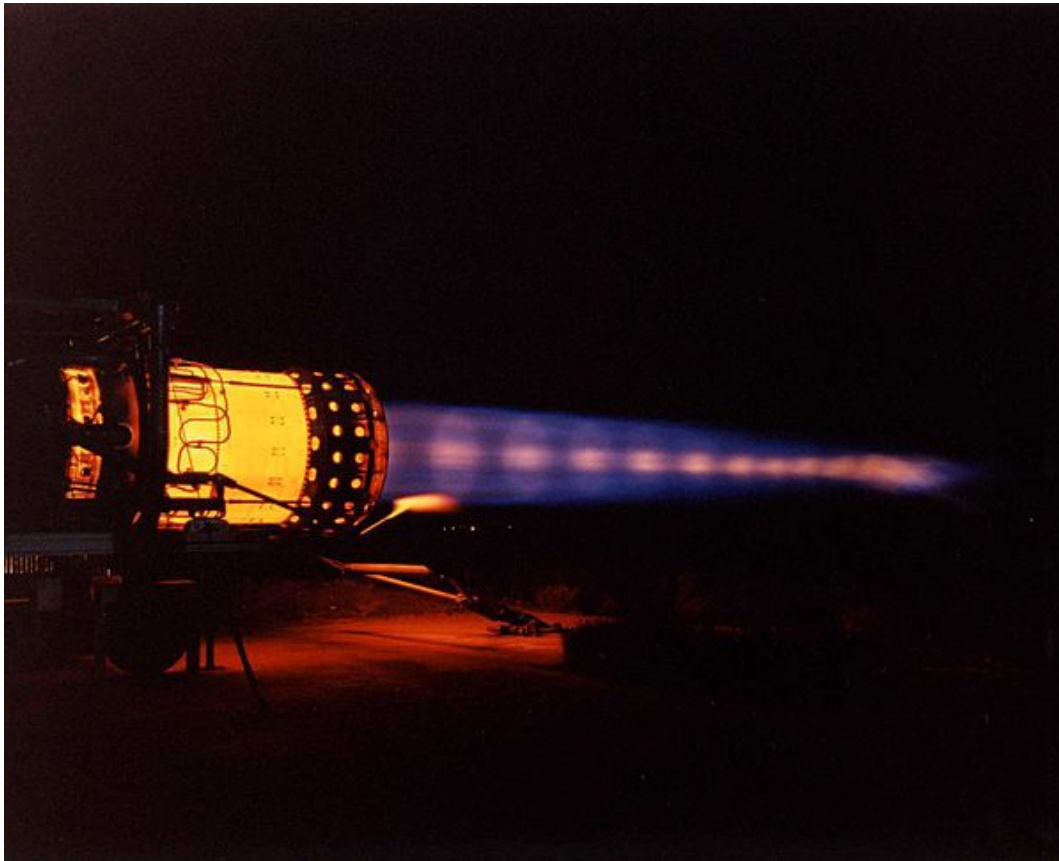

Figure 4.6: J58 Mach Diamonds from www.wikipedia.org

4.3.5   Optimal Thrust

Now we will talk about how to design a nozzle to optimize thrust. The formula for thrust is given by Equation 2.16. In Chapter 2, we were concerned with maximizing thrust by increasing the mass flow rate through the nozzle. Now we will be looking at maximizing thrust by maximizing the velocity of the flow and optimally expanding the gas.

If we have underexpanded flow, that means that we could have potentially increased the velocity of the gas by expanding it further, thus underexpanded flow doesn't maximize thrust. In overexpanded flow, since the ambient pressure is greater than the exhaust pressure, the atmosphere pushes into the nozzle, decreasing thrust (corresponds to a negative pressure thrust or a negative second term in the thrust equation). Thus, the configuration that generates the most thrust is an ideally expanded nozzle, which expands the combustion

gases to atmospheric pressure.

In the first stage of a rocket launch, the rocket has to ascend through the atmosphere into space. As the rocket gains altitude, the ambient pressure decreases, so the nozzle will tend to become underexpanded as it ascends. As a result, the nozzle is designed to be overexpanded at sea level, so as it travels upwards, it becomes ideally expanded and then overexpanded. For this reason, typical stage one engines are designed to be most efficient well above sea level.

The first stage of a SpaceX Falcon 9 contains nine engines, so in total there are nine nozzles with relatively small exit areas. In the second stage, there is one engine with a nozzle with a very large exit area. Why is this?

The first stage is designed to operate in the atmosphere of Earth, and the nozzles are designed to expand the flow to the atmospheric pressure. On the other hand, the second stage is meant to operate in space, which is almost a vacuum. As a result, the stage two nozzles need to expand the flow to a much, much lower pressure than the stage one engines. This is why the stage two nozzles have a much larger exit area.

Figure 4.7: First and Second Stage Size Comparison from Everyday Astronaut

4.4   Thrust Revisited

Now we will take a closer look at Equations 2.17-2.19 and how they are derived. First, we start with the general thrust equation, which is Equation 2.16. Then we can substitute in equation 2.15 which gives the mass flow rate for choked flow.

Now we need an expression for the exit velocity of the flow as a function of the chamber pressure and exit pressure, which are known quantities. We can use Equation 2.10 where we will set p to the exit pressure, p_e. This will allow us to calculate the Mach number of the flow as a function of chamber pressure and exit pressure. Then from Equation 2.2 we can

calculate the velocity of the flow as a function of the exit temperature, which is unknown as well. However, we can then apply Equation 2.11 to express the exit temperature in terms of the stagnation temperature. Now we have the exit velocity completely in terms of stagnation temperature, pressure, and exit pressure.

$$v_e = \sqrt{2T_0 R \left(\frac{\gamma}{\gamma - 1}\right) \left[1 - \left(\frac{p_e}{p_0}\right)^{\frac{\gamma-1}{\gamma}}\right]} \tag{4.8}$$

Now we can combine this equation and Equation 2.15 (mass flow rate) to get an equation for thrust, which is Equation 2.17. I went over this derivation just because the thrust equation is very important, so you would benefit from knowing where it comes from.

I will rewrite the thrust equation here for convenience.

$$F_{thrust} = p_0 A^* \sqrt{\frac{2\gamma^2}{\gamma - 1}\left(\frac{2}{\gamma + 1}\right)^{\frac{\gamma+1}{\gamma-1}}\left[1 - \left(\frac{p_e}{p_0}\right)^{\frac{\gamma-1}{\gamma}}\right]} + (p_e - p_a)A_e \tag{4.9}$$

Again, this can be simplified by defining the thrust coefficient.

$$F_{thrust} = p_0 A^* C_f \tag{4.10}$$

Where C_f is the thrust coefficient and is defined as

$$C_f = \sqrt{\frac{2\gamma^2}{\gamma - 1}\left(\frac{2}{\gamma + 1}\right)^{\frac{\gamma+1}{\gamma-1}}\left[1 - \left(\frac{p_e}{p_0}\right)^{\frac{\gamma-1}{\gamma}}\right]} + \frac{(p_e - p_a)A_e}{p_0 A^*} \tag{4.11}$$

The thrust coefficient is a measure of how well the nozzle amplifies the thrust past the amount if the gases were forced through a duct with the cross-sectional area of the throat. This can be seen from the equation itself. The product of p_0 and A^* would just yield the force if the pressure acted on just a throat sized area (essentially just poking a hole in the bottom of the combustion chamber with a hole area of the throat area), so the thrust coefficient gives the factor by which adding a nozzle will amplify the thrust past poking a hole.

Equation 4.9 also allows us to separate the factors that generate the thrust. If we decrease the ratio p_e/p_0, then the thrust increases. This intuitively makes sense. If the chamber pressure increases with respect to the exit pressure of the gas, the gas will be exiting at a faster velocity, so more thrust is generated. Similarly, if we increase p_0, then the density of the gas will increase, so mass flow rate and thus thrust increases. If we increase A^* and the chamber pressure remains constant and we still have choked flow, the thrust increases since the mass flow rate increases.

Equation 4.9 is also defined to experimentally determine the thrust coefficient of a nozzle. We know that the larger the pressure and throat area, the greater the thrust, but the coefficient C_f is a way to group together any non-ideal nozzle behavior such as two phase flow. Under perfectly ideal conditions, C_f is defined by Equation 4.10, but the thrust coefficient for a real nozzle will be less than this number, and measuring the thrust of the motor will allow you to experimentally determine the thrust coefficient of the nozzle.

4.5 C-Star

In this section we will look a bit more into the quantity c^* which was introduced in Chapter 2. It is defined as

$$c^* = \frac{p_0 A^*}{\dot{m}} = \sqrt{\frac{RT_0}{\gamma \left(\frac{2}{\gamma+1}\right)^{\frac{\gamma+1}{\gamma-1}}}} \tag{4.12}$$

Since the equation for c^* is a function only of R, T_0, and γ, it is purely a property of fuel combustion, and is independent of nozzle properties.

The units of c^* are m/s. This is because c^* is what the exit velocity of the gas would be if there were just a hole poked in the combustion chamber and no nozzle. This can be seen from the above equation, since the equation can be rearranged to

$$\dot{m}c^* = p_0 A^* \tag{4.13}$$

Where both sides of the equation express the thrust of the motor without a nozzle. Then it can be seen that c^* will be the exit velocity of the gas. Since the mass flow rate is proportional to throat area, c^* will be independent of throat area. This means that no matter what size hole you poke in the combustion chamber (assuming choked conditions), the gas will always have the same velocity, c^*.

Looking at Equation 4.11 and knowing what c^* physically represents, we can look at some relationships. If we increase the specific gas constant by using a lighter gas, the characteristic velocity increases. This makes sense. If we have the same thermal energy, a gas that is lighter will have a larger exit velocity. If we choose a gas with a higher combustion temperature, T_0, then the gas will have more thermal energy, and thus will have a larger characteristic velocity.

Since we know that C_f is a measure of how well the nozzle amplifies the thrust, we can easily prove that

$$v_{\text{eff}} = C_f c^* \tag{4.14}$$

Where v_{eff} is the effective exit velocity of the gas at the nozzle exit. Effective velocity is the velocity of the gas at the exit point if all the thrust was generated by a change in momentum of the gas. So effective velocity accounts for thrust generated by a change in momentum and the pressure thrust. So

$$F_{thrust} = \dot{m}v_{\text{eff}} \tag{4.15}$$

4.6 Normal Shocks

A normal shock is a discontinuity in a flow across which pressure, temperature, and density increase, and velocity decreases. For our purposes, a normal shock occurs in supersonic flow through a nozzle when pressure needs to equalize. The normal shock is always perpendicular to the incoming direction of flow (hence the name "normal"). This is a very vague and abstract definition, but we will consider examples in the next section.

A normal shock, like any shock, is not isentropic. If you have flow through a nozzle and there is a shock wave inside the nozzle, you cannot use the isentropic flow relation to get the Mach number or any property of the flow after the shock. The most important thing to remember about normal shocks, is that the flow immediate downstream of the shock is ALWAYS subsonic. This is why we never want to have normal shocks inside the nozzle. Having a normal shock inside of a nozzle is detrimental to the thrust generated by the engine.

Since the flow is decelerated across the shock, the temperature and pressure of the flow must rise. There are a set of equations that can be used to calculate flow parameters across a normal shock, and I will present them without derivation here, but they are derived by creating a control volume across the normal shock.

$$M' = \sqrt{\frac{(\gamma - 1)M^2 + 2}{2\gamma M^2 - (\gamma - 1)}} \tag{4.16}$$

$$\frac{p'}{p} = \frac{2\gamma M^2 - (\gamma - 1)}{\gamma + 1} \tag{4.17}$$

$$\frac{\rho'}{\rho} = \frac{(\gamma + 1)M^2}{(\gamma - 1)M^2 + 2} \tag{4.18}$$

$$\frac{T'}{T} = \frac{[2\gamma M^2 - (\gamma - 1)][(\gamma - 1)M^2 + 2]}{(\gamma + 1)M^2 + 2} \tag{4.19}$$

$$\frac{p_0'}{p_0} = \left[\frac{(\gamma + 1)M^2}{(\gamma - 1)M^2 + 2}\right]^{\frac{\gamma}{\gamma - 1}} \left[\frac{\gamma + 1}{2\gamma M^2 - (\gamma - 1)}\right]^{\frac{1}{\gamma - 1}} \tag{4.20}$$

$$\frac{T_0'}{T_0} = 1 \tag{4.21}$$

Where the quantities with the prime symbol are the quantities after the normal shock and the quantities without a prime are the quantities before the normal shock.

The trend that can be seen is that the faster the flow is before the normal shock, the greater each of the quantities change. This implies that the faster the flow is initially, the stronger the shock. Finally, one idea of interest is why the stagnation temperature of the flow does not decrease or even change at all. While a normal shock is not isentropic, it is still adiabatic, which means that no heat is added or removed during the process. This ensures that the stagnation temperature remains unchanged after the shock.

4.7   Nozzle Flow Cases

In this section we will look at all possible cases of nozzle flow, from no flow at all to fully supersonic ideally-expanded flow. For this analysis, we will be using the graphic shown below. It uses a slightly different naming convention. First of all, ignore the valve to the right of the image. In the combustion chamber the graphic uses p_t and T_t. These simply mean total pressure and total temperature. They are the same as stagnation pressure and temperature.

At the throat of the nozzle it says p_c, which means choked pressure. This is the pressure at the throat of the nozzle when choked flow is achieved. It can be calculated through Equation 2.10 by setting M=1, since choked flow occurs when flow is sonic at the throat and by solving for p which would be p_c. So if the pressure at the throat is ever p_c, then you know that flow through the nozzle is choked. Like usual, p_e is the pressure of the gas at its exit, and p_b is the back pressure, or ambient pressure that the nozzle is venting gas into.

The plot in the graphic shows pressure vs distance in the nozzle. However, the pressure it plots is normalized by the chamber pressure. At the rightmost portion of the graph, the pressure always has to be equal to p_b since the pressure must equilibriate. In the following subsections, we will explore how flow through the nozzle evolves as the ration p_b/p_t is lowered.
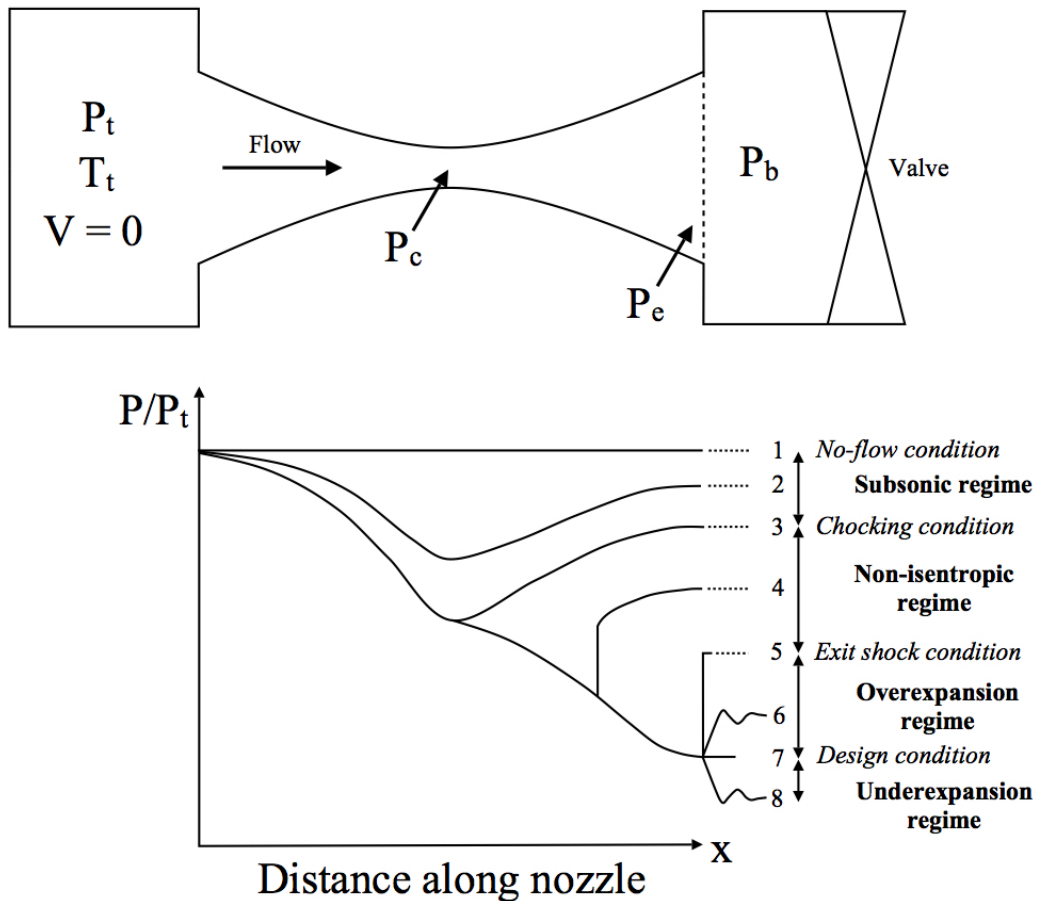

Figure 4.8: Nozzle Flow Cases from aerospaceengineeringblog.com

4.7.1   Case 1: No Flow

The easiest case of nozzle flow is no flow. This occurs when the ratio of p_b/p_t=1. This means that the chamber pressure is equal to atmospheric pressure. Since there is no pressure differential, there is no flow through the nozzle. This corresponds to line 1 on the graph.

4.7.2   Case 2: Fully Subsonic Flow

If we lower the ratio p_b/p_t either by increasing the chamber pressure or by decreasing the back pressure, we get subsonic flow through the nozzle. In this case, the flow increases in velocity in the converging section, then decreases in velocity in the diverging section. This happens because when you constrict a subsonic flow, it speeds up, and if you expand a subsonic flow it slows down. By Bernoulli's principle, the pressure of the gas in the nozzle decreases in the converging section, and increases in the diverging section. At the exit, the pressure of the flow is equal to the back pressure.

In this case, the highest velocity and lowest pressure occurs at the throat. This corresponds to line 2 on the graph.

4.7.3   Case 3: Choked Subsonic Flow

If we lower the ratio p_b/p_t further, eventually we will get to the point that the flow is accelerated in the converging section such that it is Mach 1 at the throat. However, this pressure ratio will yield an exit pressure much greater than p_c. Since the gas pressure in the nozzle must equalize with atmospheric pressure, the pressure of the gas must increase in the diverging section of the graph. Thus the flow decelerates, and by Bernoulli's principle, the pressure increases. This case is given by line 3 on the graph.

4.7.4   Case 4: Normal Shock in Nozzle

If you continue to decrease the ratio p_b/p_t, the pressure curve in the converging section will always look the same as in case 3. This is because of the same phenomena which we have looked at before where pressure information cannot be transmitted across a sonic or supersonic region. Let's consider this a bit more in-depth. Imagine we lower the ratio by decreasing the back-pressure. If we do that, the information that the back pressure is decreased cannot be sent across the throat. If we decrease the pressure ratio by increasing the chamber pressure, the pressure in the converging section will increase, but the shape of the pressure curve in the converging section will look the same, since it plots the pressure normalized by the chamber pressure.

So if you continue to decrease the pressure ratio beyond Case 3, supersonic flow starts to appear in the diverging section. This happens because the pressure differential increases further, so flow starts to get accelerated through the diverging section. However, like the previous case, the gas pressure needs to equalize with the atmospheric pressure. At this point, the flow doesn't have enough distance (across the rest of the nozzle) to isentropically decelerate to increase the pressure. As a result, a normal shock forms in order to sharply increase the pressure. Since the flow after the normal shock is subsonic, and in the diverging section, the flow continues to decelerate and increase its pressure until the exit.

If we continue to decrease the pressure ratio further, the normal shock moves from around the throat down the nozzle. Since, the flow will have more room to accelerate if the shock moves down the diverging section of the nozzle.

The condition of a normal shock in the nozzle is given by line 4.

4.7.5 Case 5: Normal Shock at Nozzle Exit

As we continue to lower the pressure ratio, the normal shock will eventually occur at the nozzle exit plane. This occurs for the same reason as in case 4. The pressure of the gas in the nozzle is well below the atmospheric pressure, so a normal shock is needed to increase the pressure of the flow. In this case, flow is supersonic throughout the diverging section of the nozzle except at the exit. This case is shown by line 5. Flow will remain to be supersonic through the entire diverging section for the remaining cases.

4.7.6 Case 6: Overexpanded

If we continue to lower the pressure ratio (see a trend here), the flow continues to be fully supersonic through the entire diverging section, and the pressure of the gas at the exit is still lower than the ambient pressure. However, this difference isn't as large as the pressure difference in case 5, so a normal shock would raise the pressure too much. As a result, a weaker oblique shock forms outside of the nozzle to attempt to equalize the pressure, however, this shock will raise the pressure a bit too much, so the flow then goes through an expansion fan to lower the pressure again. This explains the oscillating line at the end of line 6. However, as shown in the image below, the alternating oblique shocks and expansion fans are a bit more complicated than we covered. The Mach disks in red are the same Mach diamonds that that were seen in the J58 engine.
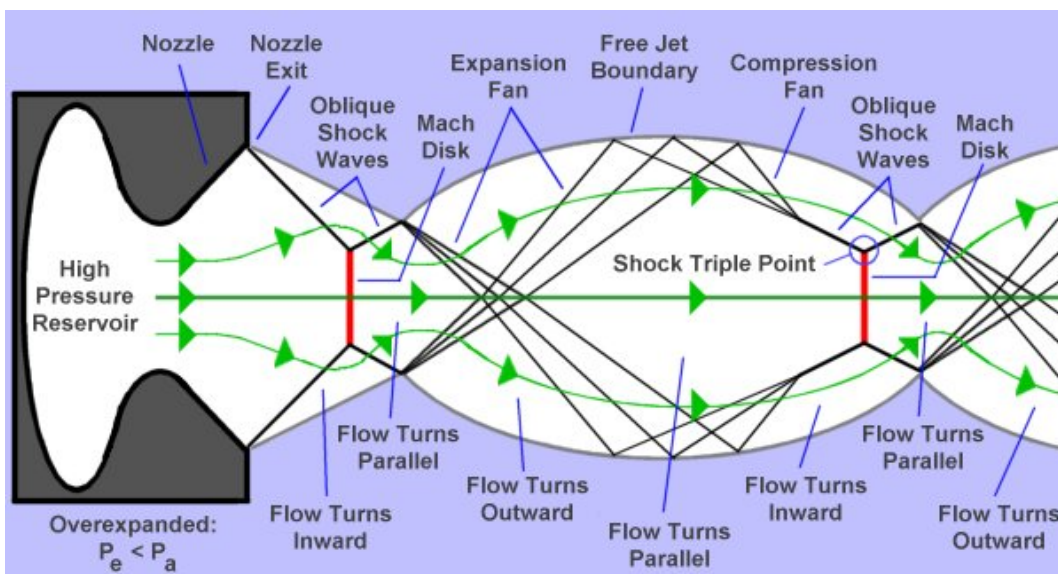

Figure 4.9: Overexpanded Nozzle Flow from aerospaceengineeringblog.com

4.7.7 Case 7: Ideally Expanded

If the pressure ratio is lowered further, we will eventually get to the case where the pressure of the gas is equal to the back pressure (ambient pressure). This case is the ideal one, where thrust is maximized. In this case, no shocks are formed. This is given by line 7.

4.7.8   Case 8: Underexpanded

If the pressure ratio is lowered further from the ideal conditions, then the pressure of the gas will be greater than the ambient pressure. Thus, the pressure of the gas needs to decrease. As a result, an expansion fan forms outside of the nozzle to decrease the pressure. However, again, it overshoots, so an oblique shock forms after the expansion fan. This accounts for the oscillating line at the end of line 8.

The pressure ratio or ranges of pressure ratios can be calculated for cases 1, 2, 3, 6, 7, and 8 by using isentropic flow relations.

4.8   Oblique Shocks

4.8.1   Basics

This section and the section on expansion fans are only important if you want to design a nozzle using the Method of Characteristics.

An oblique shock wave is another type of shock. Like a normal shock, flow passing though an oblique shock is not isentropic. Also in an oblique shock, flow is decelerated and the pressure and temperature increase. However, it differs from a normal shock in that the flow that crosses an oblique shock can still be supersonic, and usually is. This type of shock occurs when supersonic flow has to navigate an object in its flow. This shock is always slanted with respect to the incoming flow (hence the name oblique), and is always touching the object that the flow is navigating (which is why it is also referred to as an attached shock).

When any flow travels past an object, the streamlines try to align parallel to the object. Imagine subsonic flow trying to navigate a wedge placed in the flow. Since the flow is subsonic, the upstream flow is "warned" that there is an obstacle in the way, so it smoothly turns in order to remain flowing parallel to the surface of the wedge.

However, in supersonic flow, the flow upstream isn't "warned" about the obstacle. All the pressure signals that are trying to warn the upstream flow interfere constructively with each other to form an oblique shock. As soon as the flow interacts with the oblique shock, it suddenly knows that there is an obstacle in the way, so it sharply turns in order to remain flowing parallel to the surface of the wedge, as shown in Figure 4.10. A general idea that should be remembered is that streamlines always turns towards the oblique shock.
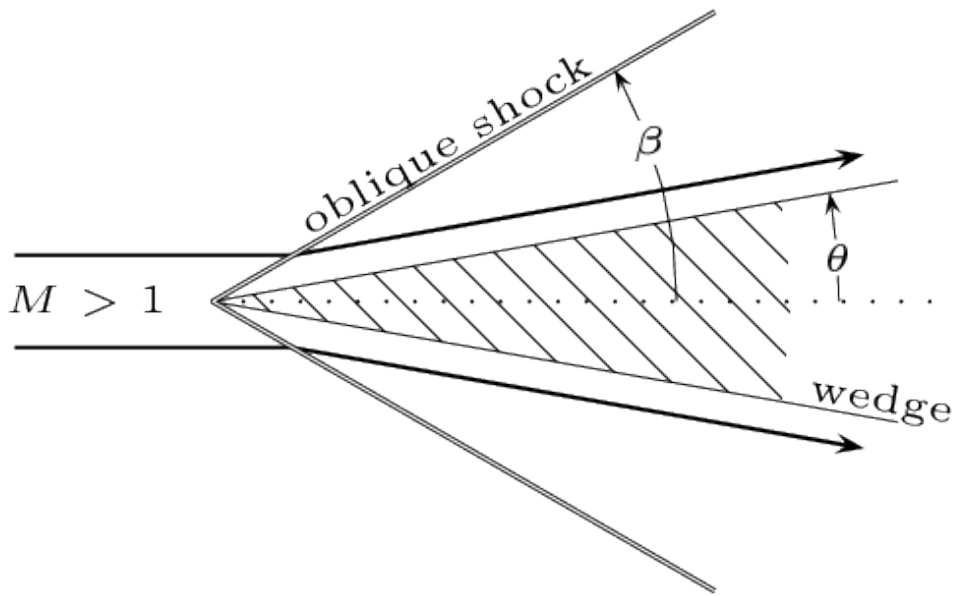
Figure 4.10: Oblique Shock from wikiwand.com

This is the same phenomena that causes oblique shocks to form with fighter jets. The jet is moving through the air at supersonic speeds, so the air upstream isn't warned of the jet's approach, but when the air gets close enough, it sharply turns in order to flow parallel to the surface of the nose of the plane. The point at which the flow suddenly turns is the oblique shock. Oblique shocks are generated whenever there is a large change in cross sectional area, so a fighter jet generates oblique shocks at its nose, wings, and elevators.
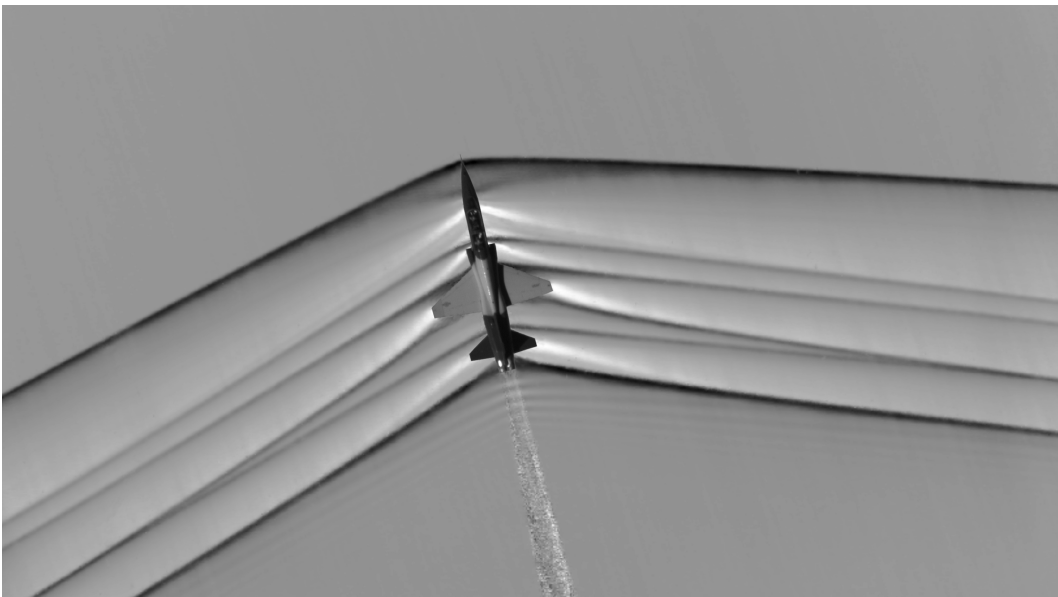

Figure 4.11: Oblique Shock due to Jet from www.wikipedia.org

4.8.2 Theta-Beta-Mach Relationship

There is a mathematical relationship between the incoming flow's Mach number, its turn angle (θ), and the angle of the oblique shock with respect to the incoming streamlines (β). This relationship is known as the theta-beta-Mach relationship. It is given by

$$tan(\theta) = 2cot(\beta)\frac{M^2 sin^2(\beta) - 1}{M^2(\gamma + cos(2\beta)) + 2} \tag{4.22}$$

This is a highly convoluted formula, so it is easier to get an intuition by looking at a graphical representation of this formula.
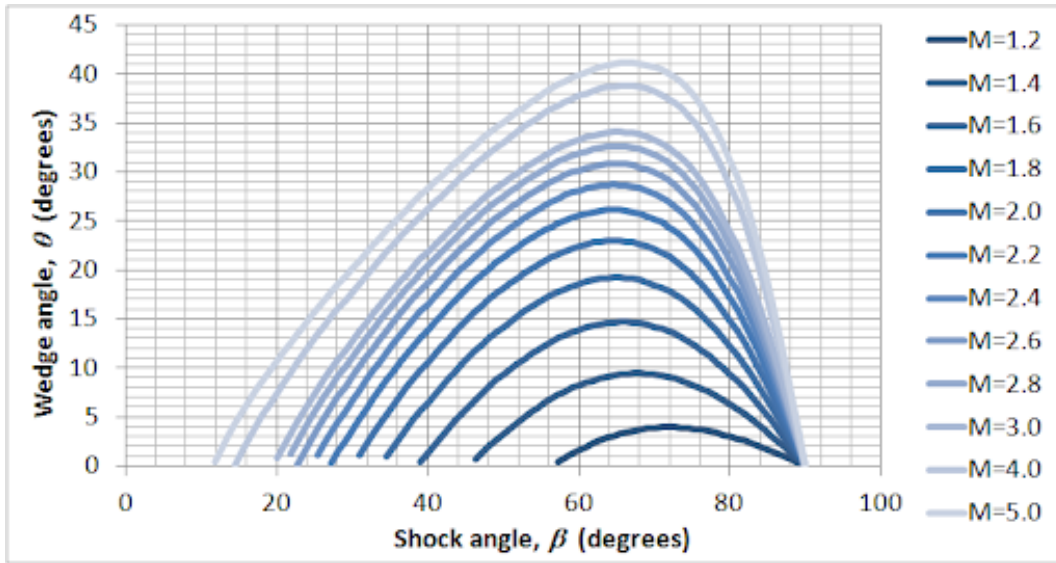

Figure 4.12: Theta-Beta-Mach Graph from flowsquare.com

A few relationships can be extracted from this graphic. If you have a wedge in a supersonic flow and the flow speeds up, then the Mach angle (β) decreases.

There exists a maximum angle that flow can be turned though by an oblique shock. This can be see by imagining a horizontal line that passes over the curve in the graph with no intersection. This means that there is no oblique shock solution for that turn angle. This doesn't mean that there is no shock present. There is a bow shock. This shock is not attached to the wedge, and can only be calculated by using Computational Fluid Dynamics simulations.

At each turn angle where an oblique shock exists, there are usually two solutions. This can be seen by imagining a horizontal line that intersects the curve. Most lines that are drawn intersect the curve twice. These two solutions correspond to a weak oblique shock at the lower angle, where flow is still supersonic after the shock, and a strong oblique shock which corresponds to the larger shock angle, where flow is subsonic after the shock. Experimentally, the weak shock is almost always the one which occurs.

4.8.3 Oblique Shock Relations

Just like normal shocks, there exist a set of formulas that can be used to quantify how the properties of a flow change as they cross an oblique shock. They will be presented here without derivation.

$$\frac{p'}{p} = 1 + \frac{2\gamma}{\gamma + 1}(M^2 sin^2(\beta) - 1) \tag{4.23}$$

$$\frac{\rho'}{\rho} = \frac{(\gamma + 1)M^2 sin^2(\beta)}{(\gamma - 1)M^2 sin^2(\beta) + 2} \tag{4.24}$$

$$\frac{T'}{T} = 1 + \frac{p'}{p} \tag{4.25}$$

$$M' = \frac{1}{sin(\beta - \theta)}\sqrt{\frac{1 + \frac{\gamma - 1}{2}M^2 sin^2(\beta)}{\gamma M^2 sin^2(\beta) - \frac{\gamma - 1}{2}}} \tag{4.26}$$

Where the quantities with primes are the properties after the shock.

4.8.4 Shock Reflection and Cancellation

An oblique shock that interacts with a wall can get reflected, and in a special case, gets cancelled and no reflection occurs.

To understand shock reflection, lets consider an example. Imagine a situation depicted in Figure 4.13. There is a wedge in a supersonic flow, so it generates an oblique shock. Now assume that this oblique shock intersects a surface like at the bottom of the picture. We know that the flow will turn towards the oblique shock as stated before. If we consider a streamline near the bottom surface, it is initially moving parallel to the surface. However, once it interacts with the shock generated by the wedge, it will turn away from the shock and turn into the surface. However, flow tries to move parallel to surfaces. The flow after the first shock is still moving supersonic, so it doesn't know that the surface is there, since the pressure signals bunch up into what becomes the reflected shock. This reflected shock then turns the flow back to being parallel with the surface.

One important thing to keep in mind is that the angle of the reflected shock is not the same as the angle of the incident shock. We know that the flow slows down after passing through the incident shock, and we know that the turn angle of the reflected shock must be equal to the turn angle of the incident shock to keep flow parallel to the surface. By the theta-beta-Mach relation, if the two flows have the same turn angles and different Mach numbers, then they must have different shock angles.
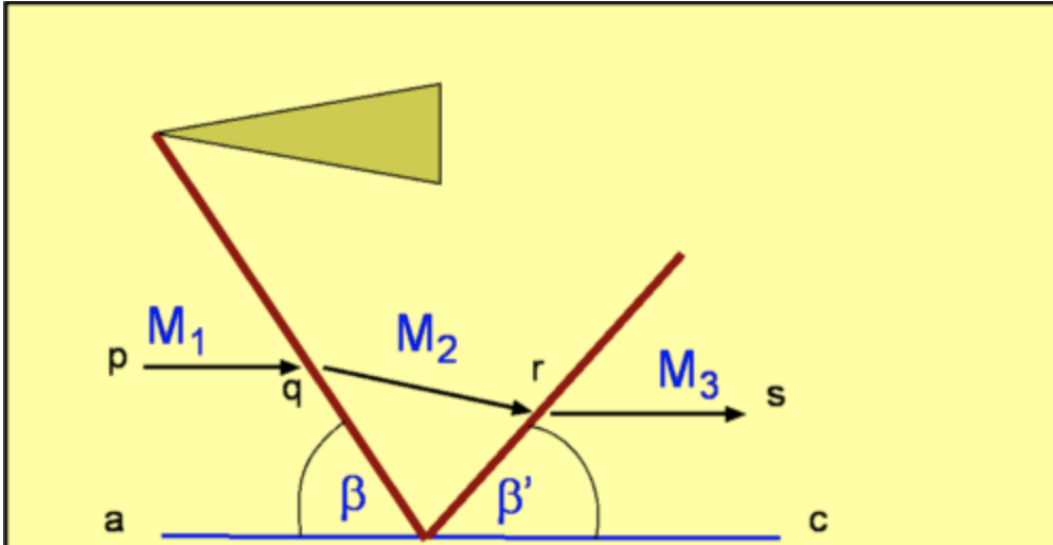
Figure 4.13: Reflected Oblique Shock from http://www.aerodynamics4students.com

Shock cancellation occurs when the surface that the shock is incident upon is inclined with respect to the horizontal at a specific angle. For this we will consider a similar example using Figure 4.14.

The reason that a reflected shock appear is because after the incident shock, the flow is not parallel to the surface. However, if the surface is inclined downwards by the same angle that the incident shock turns the flow by, no shock will appear. This is because after the incident shock, flow is already going to be parallel to the surface, so the flow does not need to be turned by a reflected shock. This idea is very, very important and is used to design nozzles using the method of characteristics.
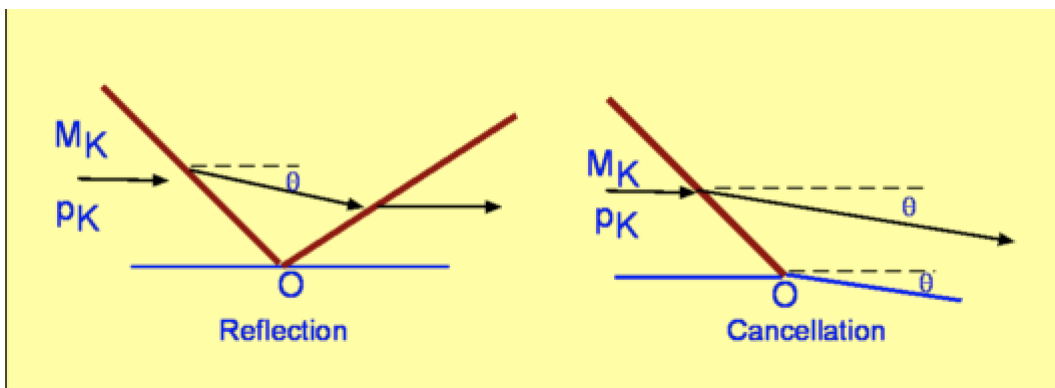

Figure 4.14: Cancelled Oblique Shock from http://www.aerodynamics4students.com

Oblique shocks will never exist inside of a nozzle, however they share many properties with expansion fans, which do exist inside nozzle, so it's important to understand how they work and their properties (like reflection, and cancellation).

4.9 Expansion Fans

Expansion Fans can be thought of as the opposite of an oblique shock in a way. Oblique shocks occur when supersonic flow has to navigate a concave object. On the other hand, expansion fans occur when supersonic flow is turned about a convex corner. Flow turns away from the expansion wave as it crosses the wave unlike oblique shocks where flow is turned towards the shock. We will look at an example to explore why this happens.

As we have seen in the last section, flow tries to remain parallel to the surfaces they flow over. When supersonic flow approaches a corner as shown in the image below, it tries to remain parallel to the flow. But in this case, flow isn't being compressed by a wedge or concave corner, it is being expanded by a convex corner. As a result, the flow speeds up and turns isentropically.
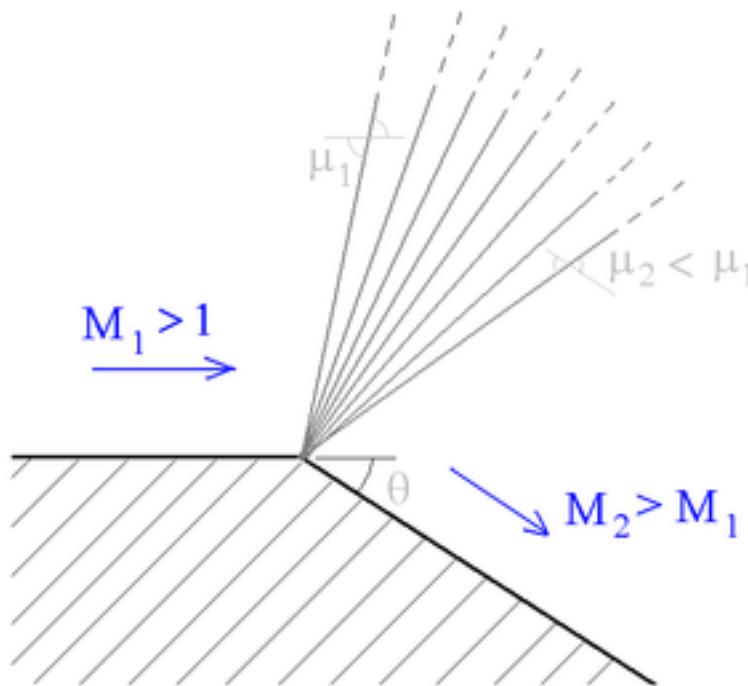

Figure 4.15: Expansion Fan from www.wikipedia.org

However, the flow isn't turned by a single expansion wave, as that would violate the second law of thermodynamics. Instead, the flow is turned by an infinite number of expansion waves, also known as an expansion fan. Each individual expansion wave turns the flow by an infinitesimal amount.

The more that a flow is turned by an expansion fan, the faster the flow becomes. This idea can be quantitatively expressed by the Prandtl-Meyer Function. This function tells you how much a sonic (M=1) flow has to turn in order to reach a certain Mach number.

$$\nu(M) = \sqrt{\frac{\gamma+1}{\gamma-1}} arctan\sqrt{\frac{\gamma-1}{\gamma+1}(M^2-1)} - arctan\sqrt{M^2-1} \qquad (4.27)$$

So if you have a sonic flow and you want to see how much you have to turn the flow to

accelerate it to Mach 2, you just have to plug in 2 for M and evaluate to get the turn angle necessary.

If you have an incoming flow that is greater than Mach 1, you can still calculate the turn angle needed for it to reach a certain Mach number. You just have to use the formula

$$\nu(M_2) = \nu(M_1) + \theta \tag{4.28}$$

It is possible for an expansion fan to not be able to turn a flow past a certain angle. This is similar to how an oblique shock has a maximum turn angle for certain flows. To get the maximum turn angle for an expansion fan, you need to plug in infinity for M.

Expansion waves are reflected by surfaces just like how oblique shocks are. The reflected expansion wave is there to turn the flow back to being parallel to a surface. This is easy to calculate if you only have one expansion wave, but if you have an expansion fan being reflected, the reflected expansion waves are bent by the incident waves. This is not easy to calculate, but we will consider how to do this when using Method of Characteristics.
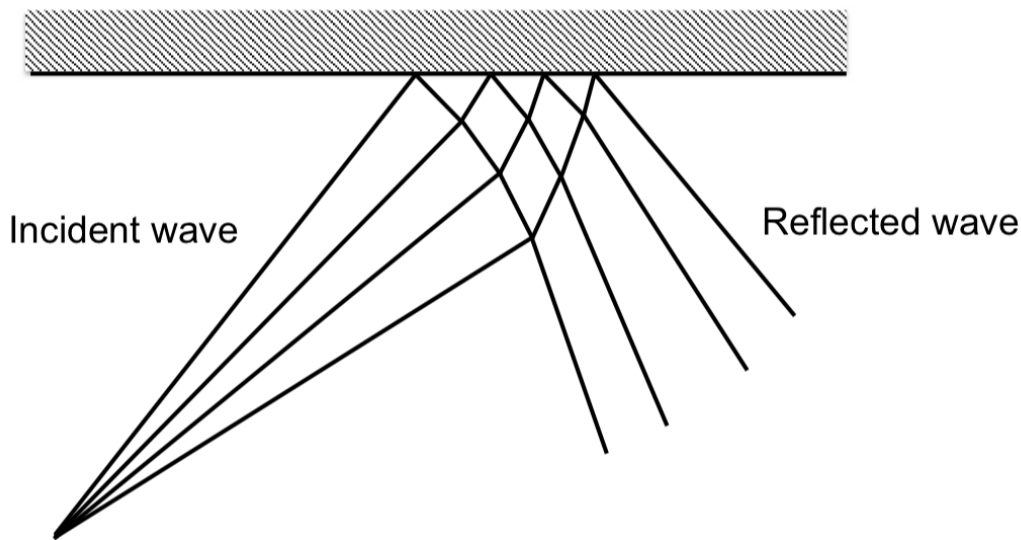

Figure 4.16: Expansion Fan Reflection from Université de Liège: Aerothermodynamics of High Speed Flows

Similar to oblique shocks, expansion waves can be cancelled by having the surface slanted at the same angle and in the same direction that the incident expansion wave turns the flow. This idea is very important when designing a nozzle using Method of Characteristics.

4.10   Two-Phase Flow

Up to this point, we assumed that the combustion product is only a gas. In real motors this is not the case. There are small solid particle in the exhaust which comprise the smoke coming out of the nozzle. These solid particles create a few problems.

Firstly, since they are solid, they cannot be expanded like a gas can. Two other phenomena, known as thermal and velocity lag, also affect solid particles. Thermal lag refers

to the fact that the solid particles do not transfer heat to the gas as quickly as gas would transfer heat to other gas. This results in thermal energy being thrown out of the engine without it being used to heat up and accelerate the gas. Velocity lag refers to the fact that solid particles cannot be accelerated at the same rate that a gas can. As a result, the solid particles that exit the nozzle move slower than the gas exiting the nozzle.

These factors can be accounted for by generating a new specific "gas" constant and a new ratio of specific heats. The new specific "gas" constant is defined as the universal gas constant divided by the molar mass of the gas/solid mixture. Calculating the adjusted ratio of specific heats is a bit more complicated and is shown in its entirety on nakka-rocketry.net.

4.11   Conclusion

The nozzle accelerates the gas from the combustion chamber simply by varying the cross-sectional area that the gas flows through. To maximize thrust, the nozzle should expand the gas from chamber pressure to atmospheric pressure.

Chapter 5

Nozzle Design

A nozzle is the most efficient when it expands the gas from the chamber pressure to atmospheric pressure. This ideal expansion ratio can be calculated via isentropic flow relations. You must already know what chamber pressure and throat area you will use. This was covered in Chapter 3. First use Equation 2.10 where p_0 will be the chamber pressure and p will be set to the ambient pressure. From this, you can extract what the Mach number the flow should be at the exit. Then you use Equation 2.13 and plug in the exit Mach number for M and solve for A, which will yield what exit area your nozzle should have. To start designing the nozzle, you must know the throat area and the exit area (which was covered in Chapter 3).

5.1 Conical Nozzle

A conical nozzle is the easiest to design and manufacture. To design the nozzle contour, you draw two straight lines. One forms the converging section, and one becomes the diverging section as shown below.
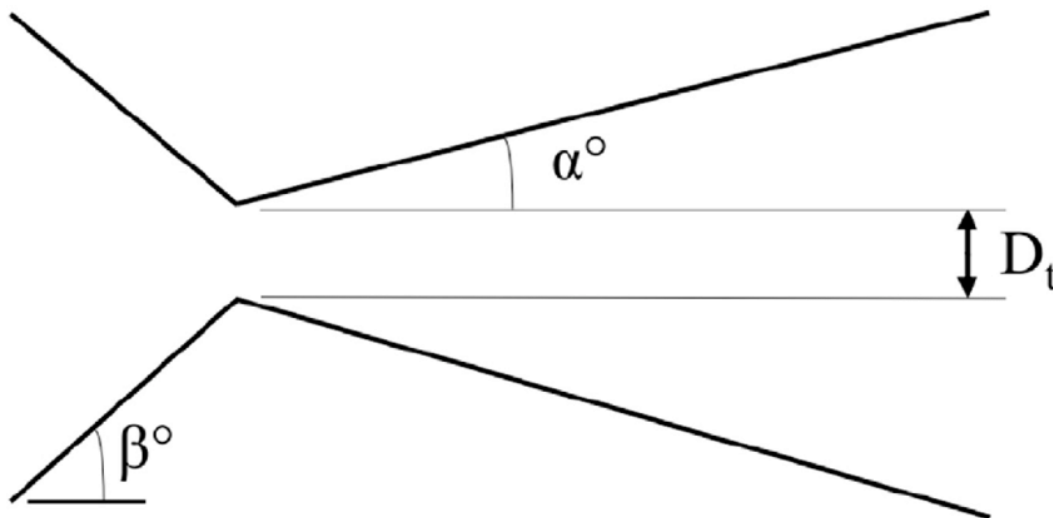

Figure 5.1: Conical Nozzle from www.researchgate.net/

The final nozzle cross section should look like Figure 5.1, but the converging and diverging section should not join at a point. This sharp corner could cause flow separation from the wall of the nozzle leading to a shock, drastically lowering performance. This corner should be filleted.

Your throat area and exit area are fixed, but you have freedom to choose the converging half-angle (β) and the diverging half-angle (α). The problem with conical nozzles is that as gas flows out of the nozzle, it isnt directed purely axially. It has some exit angle. This angle results in thrust losses. If you want to decreases these losses, you need to decrease the divergent half-angle in order for your flow to exit more axially. However, by decreasing the divergent half-angle, the nozzle has to be longer, and thus heavier, so these two effects have to be balanced.
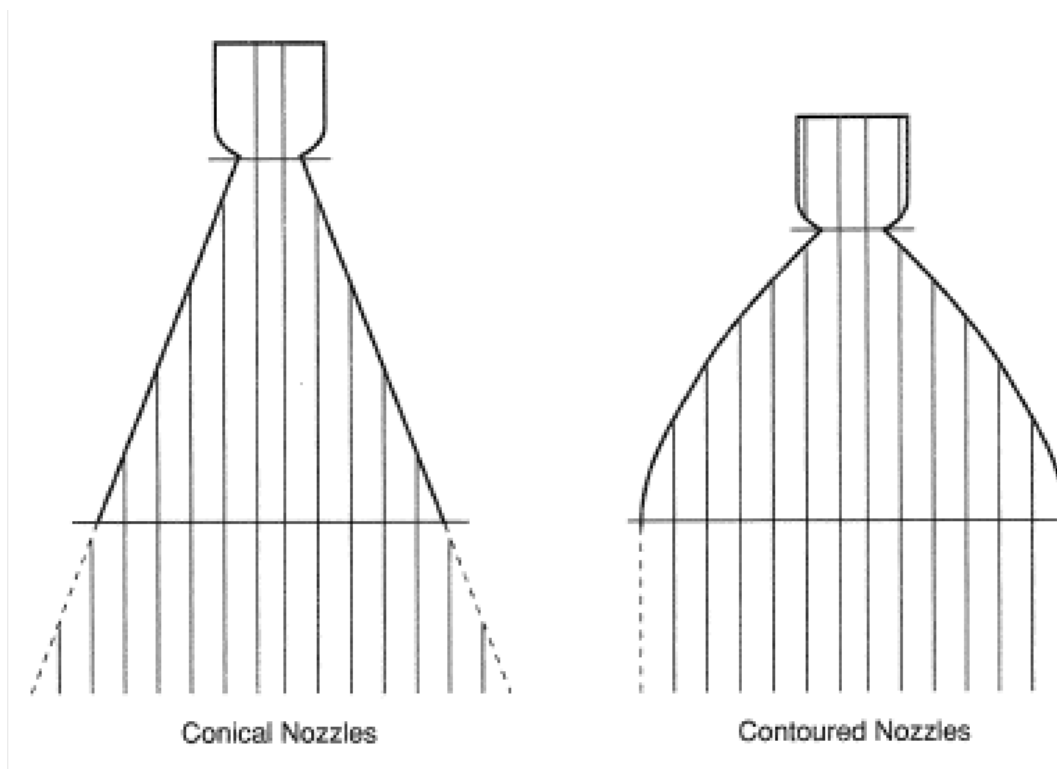

Figure 5.2: Conical Nozzle Thrust Losses from thermopedia.com

Typically conical nozzles are designed with a convergent half angle of 30 degrees and a divergent half-angle of 15 degrees.

5.2   Method of Characteristics

In a nozzle, designing the converging section isn't too difficult. You just need to create the converging section without any sharp turns that could cause flow detachment. The diverging section is much harder to design, due to the expansion fan generated when the flow initially expands at the throat.

Method of Characteristics (MOC) is a way of designing the most efficient diverging section

of a nozzle. No other shaped nozzle will result in more thrust. Since flow is sonic at the throat and the diverging section is essentially a convex corner, when flow moves past the throat, expansion waves are generated. In a conical nozzle, these expansion waves reflect off of the wall of the nozzle and into the exhaust. This results in a loss in thrust. A nozzle designed by MOC has the walls contoured in a way that they cancel the expansion waves. This nozzle also directs flow to be perfectly axial at exit unlike conical nozzles.

5.2.1   Mach Wave and Expansion Wave Reflection

A Mach wave is the small pressure wave than an object gives off to signal its presence in a flow. When flow is supersonic, these Mach waves stack up to create shockwaves or expansion waves depending on what type of corner the flow is turned by. When no object is placed in the flow, this can be modeled as an infinitely thin object being placed in the flow. The angle of the Mach waves produced is given by

$$\mu = arcsin\left(\frac{1}{M}\right) \tag{5.1}$$

This can be proven by plugging in $\theta = 0$ into the theta-beta-Mach relationship. When you have a supersonic flow, two Mach lines are generated: one with angle μ with respect to the flow direction, and one with angle $-\mu$ with respect to the flow direction. This is the same thing that happens with oblique shocks as well (since oblique shocks are a form of Mach wave). When a wedge is placed in a supersonic flow, two Mach waves are generated symmetrically about the wedge. One going upwards, and one going downwards. Oblique shocks and expansion fans are a form of Mach waves.

If we look back at Figure 4.15 we can use Equation 5.1 to calculate the angles of the expansion fan. If we use the formula we get

$$\mu_1 = arcsin\left(\frac{1}{M_1}\right) \tag{5.2}$$

and

$$\mu_2 = arcsin\left(\frac{1}{M_2}\right) \tag{5.3}$$

These angles are defined with respect to the direction of the incoming flow, so μ_1 is defined with respect to the horizontal, and μ_2 is measured with respect to to θ.

5.2.2   MOC Theory

Method of Characteristics is a way of determining properties of a flow by making use of characteristic lines. Characteristic lines are Mach lines of a flow. From every point in the supersonic flow, there are two Mach lines that pass through that point, a left running characteristic line and a right running characteristic line. An observer on a left running characteristic line looking downstream will see the line drift to the left, and an observer on a right running characteristic will see the line appear to drift to the right. In the graphic below, ignore the S+ and S-.
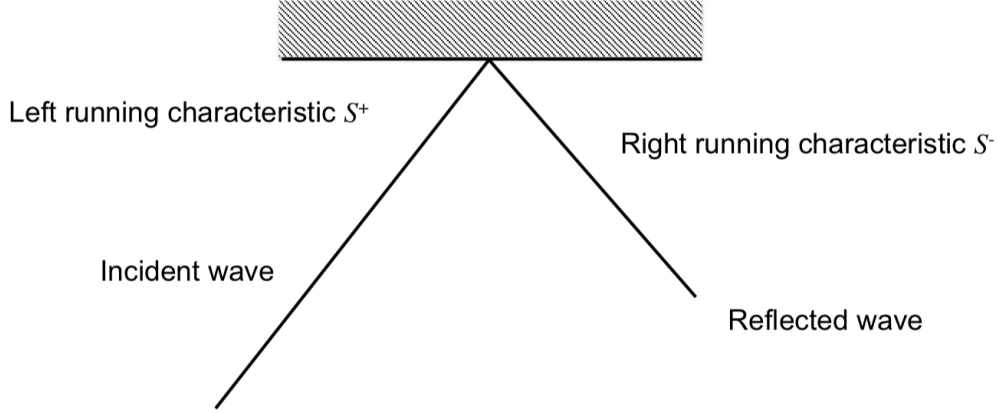
Figure 5.3: Characteristic Lines from Université de Liège Aerothermodynamics of High Speed Flows

The left running characteristic line has an angle of $\theta + \mu$ with respect to the horizontal, where θ is the angle that the flow is travelling in with respect to the horizontal, and μ is the local Mach angle. The right running characteristic line has an angle of $\theta - \mu$ with respect to the horizontal.

On each characteristic line, a certain quantity remains constant.

For a right running characteristic, the following quantity remains constant

$$CR = \theta + \nu(M) \tag{5.4}$$

For a left running characteristic, the following quantity remains constant

$$CL = \theta - \nu(M) \tag{5.5}$$

Where $\nu(M)$ is the Prandtl-Meyer (PM) angle of the flow at that point, and θ is the angle the flow makes with the horizontal at that point.

Thus, if we are looking at the intersection point of a right running and left running characteristic line, we can determine the angle of the flow, and the Mach number of the flow (since we know the PM angle), by solving Equations 5.4 and 5.5 simultaneously.

At the intersection point of a right and left running characteristic, the flow angle and the PM angle are given by the following two equations

$$\theta = \frac{CL + CR}{2} \tag{5.6}$$

$$\nu(M) = \frac{CL - CR}{2} \tag{5.7}$$

When performing this method, we can analyze the top half of the nozzle contour, since the nozzle is symmetric. The point of this method is to plot the trajectory of the expansion waves and then design the slope of the nozzle wall with the same slant angle as the angle that the flow is turned by in order to cancel out the incident expansion wave. We have to use MOC, because as expansion waves interact with each other, they bend, and this phenomena must be accounted for via MOC.
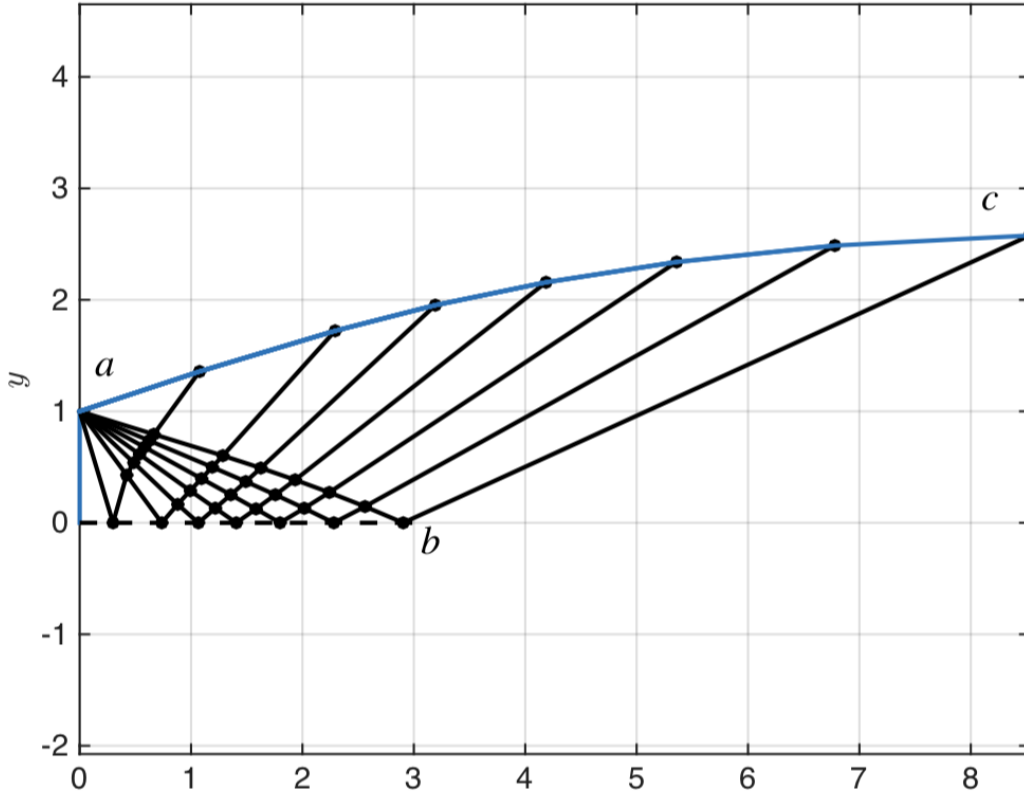
Figure 5.4: MOC Example from Université de Liège Aerothermodynamics of High Speed Flows

In this method, the maximum wall angle, θ_{max} occurs at the throat of the nozzle. The expansion waves generated by the throat look like they are reflected off of the center-line. However, these reflected waves are the expansion waves generated by the other half of the nozzle. The expansion waves generated by the throat are right-running characteristic lines and the reflected expansion waves are left-running characteristics.

At point c in Figure 5.4, the flow has the design Mach number, M_e (which can be calculated using A^* and A_e) and flow angle $\theta = 0$, since flow is purely axial at the nozzle exit. The flow at point b has $\theta = 0$, since it is on the center-line of the nozzle. Since point b is on the same characteristic line as point c and they have the same flow angle, they also have the same Mach number.

Flow is initially turned by θ_{max} as it passes through the throat. Then it is turned back by θ_{max} in order to flow axially though the nozzle exit. Thus, the flow is turned in total by $2\theta_{max}$ in the nozzle. Since the flow will be turned by $\nu(M_e)$ in total, we can see that

$$\theta_{max} = \frac{\nu(M_e)}{2} \tag{5.8}$$

We can choose the number of characteristic lines originating from the throat of the nozzle. The more characteristic lines we choose, the more accurate our designed nozzle will be. Each characteristic line will turn the flow by $\Delta\theta = \frac{\theta_{max}}{n}$ if we have n characteristic lines.

On the jth characteristic line coming from the throat, we know that the turn angle will

be

$$\theta_j = j\Delta\theta \tag{5.9}$$

Since the flow is initially sonic, we know that on the jth characteristic line the PM angle is

$$\nu(M)_j = j\Delta\theta \tag{5.10}$$

Since the PM angle is the amount that flow had to turn to reach the Mach number it is currently at, and we know how much the flow did turn already, we know that the turn angle must be equal to the PM angle.

Since we know the turn angle and PM angle for each characteristic line, we know the left and right characteristic constants for each of the characteristic lines.

$$(CL)_j = 0 \tag{5.11}$$

$$(CR)_j = 2\theta_j \tag{5.12}$$

Since all of the characteristic lines forming from the throat are right running, they have slopes

$$m_j = tan(\theta_j - \mu_j) = tan(\theta_j - arcsin(1/M_j)) \tag{5.13}$$

If you know the throat radius (which you do), you can plot the incident expansion waves since they all pass through the point (0,Rt) (where Rt is the radius of the throat) and have slopes as shown in Equation 5.13.

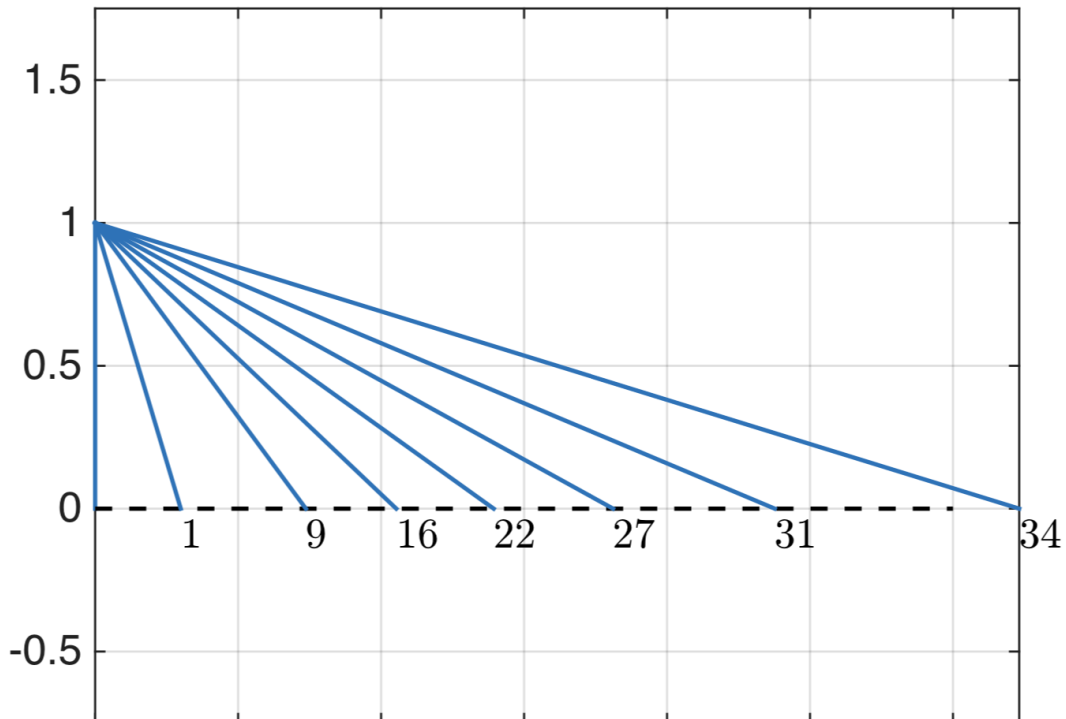Now that we have the incident expansion waves, we have to plot the "reflected" expansion waves and how they interact with the incident waves.

We know intuitively that on the center-line the flow angle should be zero degrees (since flow will be purely axial). This is true for all of the nodes on the center-line but the first. We assume that the flow angle at the first node is a small angle ($\Delta\theta$), otherwise, we wouldn't be able to start the algorithm. Since we know the turn angle and PM angle at node 1 (since we know CR), we can calculate the left running characteristic constant and then its intersection node with the characteristic line passing from 1 to 9. We know the characteristic constants of both characteristic lines at this intersection node, so we can calculate the turn angle, PM angle, and Mach number at this intersection node, which is node 2 in Figure 5.6.

Now we can create a new characteristic line coming from node 2. Since we know the properties at node 2, we can get the slope and the left-running characteristic constant of this new line (from 2 to 3). Then we can get the information at node 3 since we know the characteristic constant of the characteristic line from node 1 to node 16. We can repeat this process until we find where the reflected characteristic line intersects the last incident characteristic line. Now we are ready to calculate the first coordinate of the wall.

Since we know the Mach number and and turn angle at node 7, we know the slope of the left running characteristic line from node 7 to the first wall node. The wall can be modeled as a straight line with slope θ_{max} starting at the throat. The intersection of these two lines is the coordinate of the first wall node. The mach number and flow angle are the same at node 7 and the first wall node (node 8).
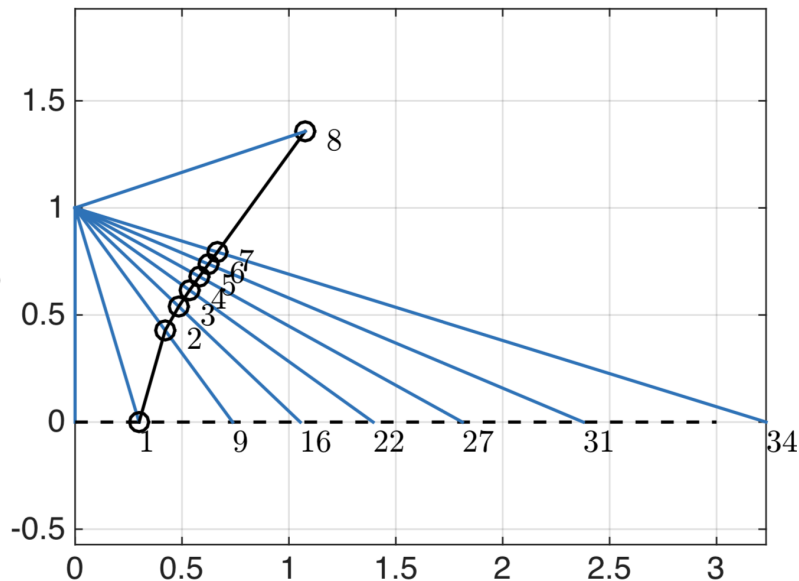

Figure 5.6: First Reflected Characteristic Line from Université de Liège Aerothermodynamics of High Speed Flows

To get the second reflected characteristic, we follow the same procedure that we used to

get the trajectory of the first characteristic line. However, this time the flow deflection angle at node 9 will be 0, since it is on the center-line. This means that the PM angle at node 9 is the same as the right-running characteristic constant from the throat to node 9. This time when we calculate the coordinates of the wall point, we extend a line with angle θ_8 from point 8. This ensures that the angle of the flow at the wall, which is θ_8, is the same as the angle of the wall, so the reflected expansion wave is cancelled.
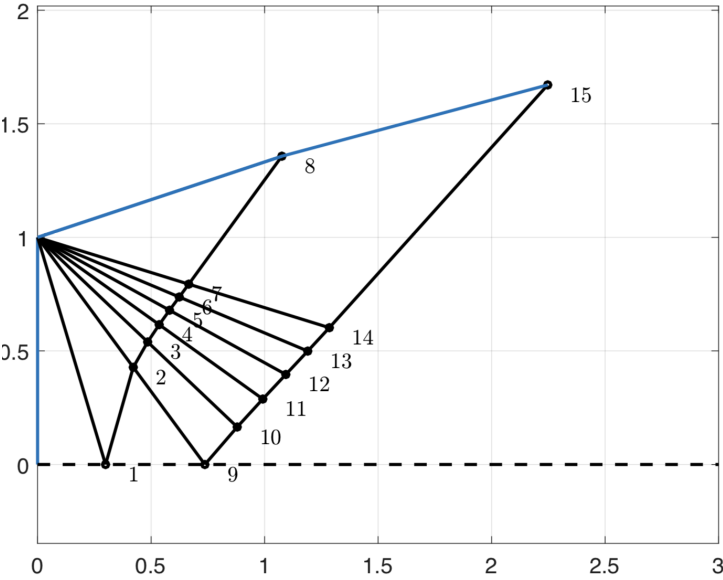

Figure 5.7:   First Two Reflected Characteristic Lines from Université de Liège Aerothermodynamics of High Speed Flows

We then follow this same procedure to calculate the rest of the wall coordinates. When we finish, we will get a plot shown in Figure 5.8.
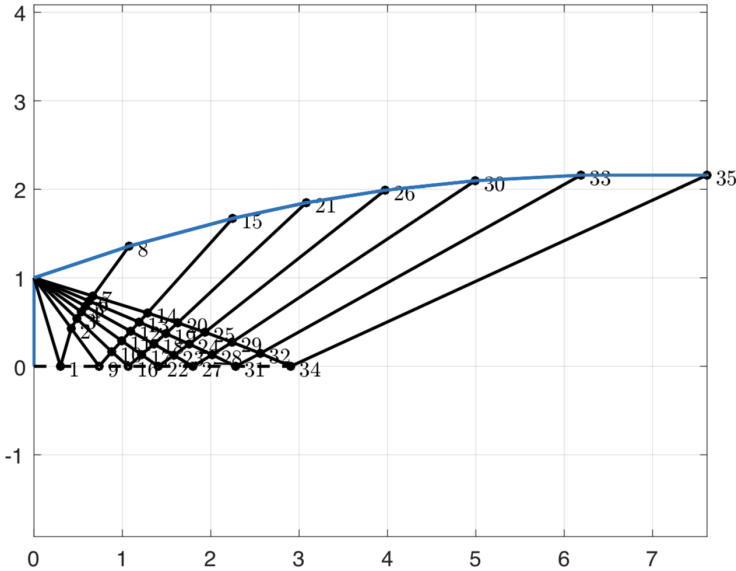

Figure 5.8:   Completeted MOC Nozzle Contour from Université de Liège Aerothermo-

dynamics of High Speed Flows

5.2.3   Designing MOC Nozzle in Practice

Designing a nozzle like this in practice would involve writing a script in Matlab. You can use any language, but it is easiest to do so in Matlab.

The four parameters that this script takes in are the throat radius, the exit Mach number, the ratio of specific heats of the gas, and the number of characteristic lines that will be used.

When I made this program, I created four helper functions. The first was a PM function, where you inputted the Mach number of the flow and the ratio of specific heats and it outputted the PM angle. The second was an inverse PM function, where you input the PM angle of a flow and it tells you the Mach number of the flow. The third was a Mach angle function, where you inputted the Mach number of a flow and it would tell you its Mach angle. The last function was the most complex. If you inputted the maximum turn angle of the flow θ_{max} and the number of characteristic lines n, it would output six vectors: a vector of the PM angles at each node, a vector of the right running characteristic constants at each node, a vector of the left running characteristic constants at each node, a vector of the flow angles, θ at each node, a vector of the Mach numbers at each node, and a vector of the Mach angles at each node.

The general structure of the overall code is firstly calculating the maximum wall angle, by using Equation 5.8. Then it calculates how many nodes there will be depending on the number of characteristic lines that will be used. The formula is

$$nodes = \frac{n(n+1)}{2} + n \qquad (5.14)$$

From there, the code uses the fourth function mentioned above to calculate the PM angles, the left and right running characteristic constants, the flow angles, the Mach numbers, and Mach angles. All of these are calculated without knowing the coordinates of the nodes. Finally, the coordinates of the nodes and walls are calculated, the characteristic lines are plotted, and the coordinates of the wall are plotted and connected to make the nozzle contour.

In this section I will go into detail as to how the fourth function works and how the coordinates of the nodes and wall are are calculated. These two are the hardest parts of the program.

The program calculates all the nodal properties by analyzing the nodes in numerical order as shown in Figure 5.8. So first it calculates the properties of node 1, then node 2, and so on.

We know that for the right running characteristics from the throat to 1, from the throat to 9, and so on until the throat to 34, the characteristic constant is $2\theta_j$ (from previous section on theory). When we consider node 1, we assume that the flow angle at node 1 is $\Delta\theta$, or else the method won't start. Since we know its flow angle and its CR, we can figure out its PM angle, which is also $\Delta\theta$. Now we can calculate CL for the line 1-2, which is 0. Since node 2 is the intersection between two characteristic lines than we know the constants of, we can figure out all the properties at node 2 by using Equations 5.6 and 5.7. By using this

51

same procedure, we can calculate the values at the rest of the nodes on the first reflected characteristic.

Now we need to calculate the properties at the first wall node. It can be shown that the flow angle at node 7 is θ_{max} (by Equation 5.9), which also must be the flow angle at the first wall node, since the wall has angle θ_{max} from point 1 to 8. Since nodes 7 and 8 are on the same characteristic line and they have the same flow angle, all their properties can be shown to be identical. This is also true for all wall nodes. The data at a wall node is exactly the same as the data at the previous node.

For all of the center-line nodes except for the first one, the flow angle is zero. Since they are all on the same right running characteristics as the node diagonally to the left and up, we know that for the center-line nodes $\nu(M) = CR$ where CR is the right-running characteristic constant for the characteristic line coming from the node diagonally up and to the left. For example, if we are analyzing node 9 in Figure 5.6, its right-running characteristic constant will be equal to node 2's CR.

Now that we know the flow angle and the PM angle at the center-line node, we can determine the rest of the properties at these nodes such as Mach number and Mach angle through the helper function. Then we can calculate their left running characteristic constant (which will be $2\nu(M)$). Now we can calculate all the values at the rest of the interior nodes. For example, to calculate the data at node 10, we use CL from node 9 and CR from node 3. We repeat this process until we calculate the data at each node.

Now that we have the data at each of the nodes, we just need to find the location of each of the nodes. We know one coordinate already (0,Rt), which is the location of the throat. We can then use equation 5.13 to calculate the slopes of the incident characteristic lines. Their intersection points with the x-axis gives the coordinates of the center-line nodes. We can use the following equation to calculate the slopes of the left running characteristics from the jth node

$$m_j = tan(\theta_j + \mu_j) \tag{5.15}$$

We will go over a quick example on how to use this formula. We already know the slopes of the incident characteristic lines and can develop an equation for these lines. To calculate the coordinates of node 2, we need to find the intersection point between the line from the throat to node 9 (which we know) and a the left-running characteristic from node 1. So we just need to get an equation for the latter. We know the coordinates of node 1, so this serves as our point, and we can use Equation 5.15 to get the slope of the line extending from node 1. Now we solve a system of equations to get the intersection point that becomes node 2. Now we use the coordinates of node 2 as the point and use the data at node 2 and Equation 5.15 to get the slope of the refracted left characteristic. Then again we solve the system of equations to get the coordinates at node 3. We repeat this process to get the coordinates of all the interior nodes.

To get the coordinates of the first wall node, we find the intersection point of the left characteristic line from node 7 and a line with angle θ_{max} from the point (0,Rt). To get the coordinates of the second wall node (node 15), we find the intersection point between the left characteristic line from node 14 and a line from the first wall point (node 8), with angle θ_8. This is to ensure that the slope of the wall is the same as the flow angle, so the reflected

Mach line is cancelled by the wall.

In this section, I used a numbering convention assuming that 7 characteristic lines were chosen, since they matched the figures shown. In the program, the number of characteristic lines will be an input and the program must account for this.

Although a MOC nozzle is the most efficient nozzle at generating thrust, it tends to be very long and thus heavy. Two solutions are available for this: you could truncate the nozzle, or you could use Rao's Method, which will be described in the next section.

5.3   Rao's Method

Rao's method for nozzle design is a method that is meant to closely imitate the geometry of the MOC nozzle, while making it shorter, and thus lighter. For this reason, Rao's nozzles are considered the standard nozzle design. This is independent of the ratio of specific heats of the gas (unlike the MOC nozzle). The diverging portion of the nozzle is generated by a quadratic Bezier curve (a curve defined by control points) which makes it look like a bell, so this nozzle is also referred to as a bell nozzle.

The length of Rao's nozzles are measured as fractions of the length of a standard 15 degree diverging half-angle nozzle. The length of the diverging section of a 15 degree conical nozzle is

$$L_{conical} = \frac{(\sqrt{\epsilon} - 1)R_t}{tan(15)} \tag{5.16}$$

Where ϵ is the expansion ratio and R_t is the throat radius of the nozzle. This formula can be verified by using a bit of trig.

An 85% bell nozzle has a diverging section with a length of $0.85L_{conical}$. Lower percent bells will result in a shorter, lighter nozzle, and larger percent bells will result a higher efficiency, but heavier nozzle. These two factors must be balanced. Usually 85% bell is chosen.
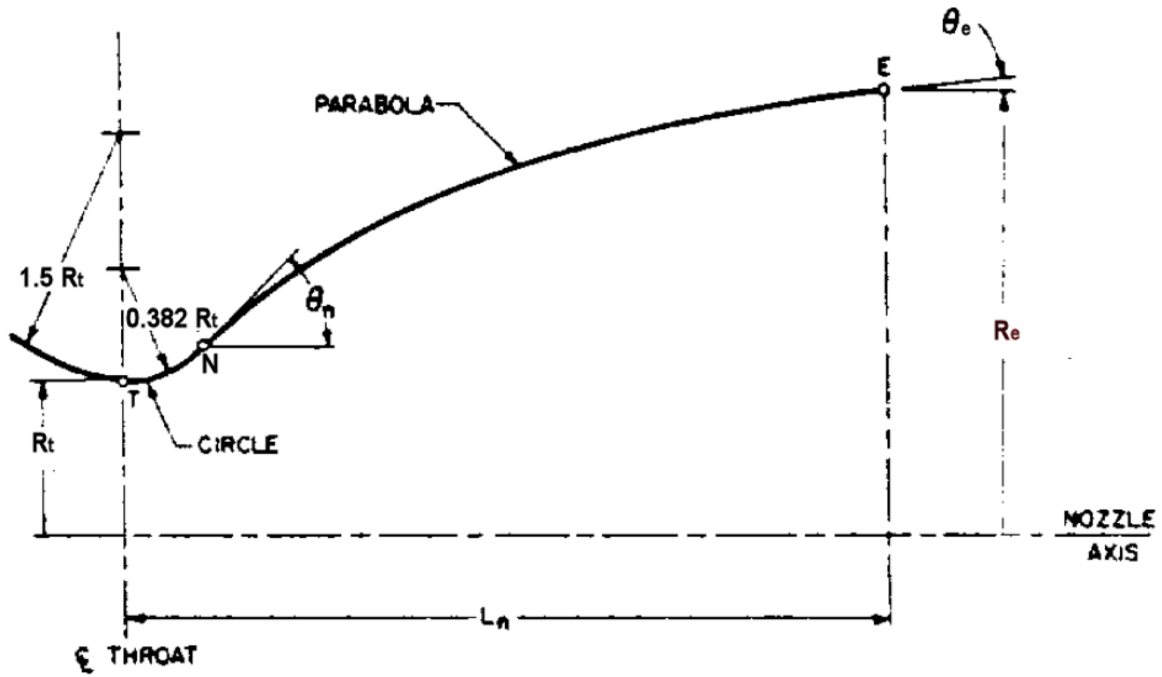
Figure 5.9: Geometry of Rao's Nozzle from http://www.aspirespace.org.uk

Figure 5.9 shows the general geometry of Rao's nozzle. The parameters θ_n and θ_e are chosen based on the expansion ratio and percent bell of the nozzle. They are selected from the chart shown below.
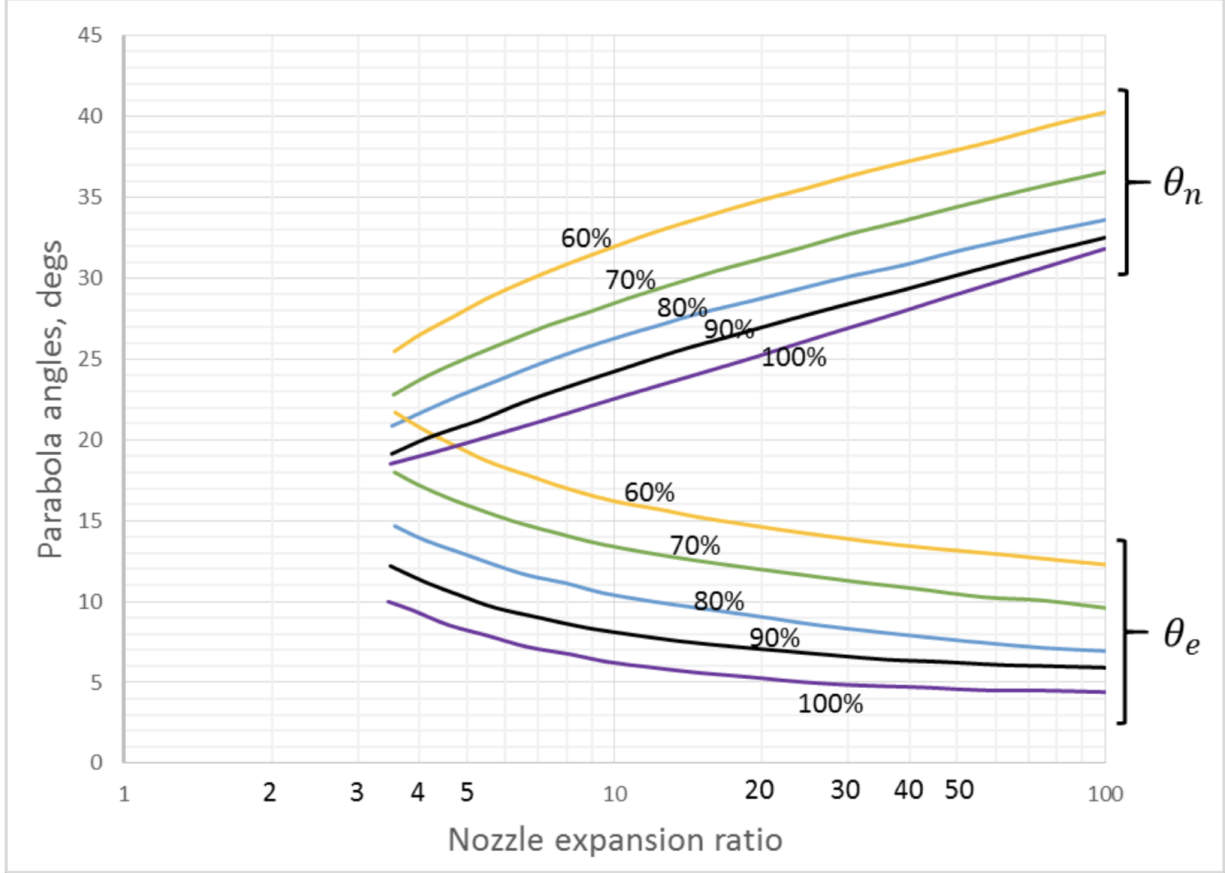
Figure 5.10: Angle Chart for Rao's Nozzle from http://www.aspirespace.org.uk

The design of the nozzle contour consists of three parts: two circular arcs, and the Bezier curve. First, an arc with radius $1.5R_t$ is drawn from -135 degrees to -90 degrees, where the angle is measured from the +x-axis. The end point of this arc is the throat of the nozzle (the x-position of the throat of the nozzle is the zero point on the x-axis). Then a curve with radius $0.382R_t$ is drawn from -90 degrees (connected to the throat) to $\theta_n - 90$ degrees. The endpoint of this second arc is point N. A a quadratic Bezier curve is then drawn from point N to point E. Point E is the exit point of the nozzle. Point E has an x-coordinate of $(0.85L_{conical})$ for an 85% bell nozzle, and a y-coordinate of $\sqrt{\epsilon}R_t$.

The parametric equations for the quadratic Bezier curve are as follow

$$x(t) = (1 - t^2)N_x + 2t(1 - t)Q_x + t^2E_y \tag{5.17}$$

$$y(t) = (1 - t^2)N_y + 2t(1 - t)Q_y + t^2E_y \tag{5.18}$$

$$0 \le t \le 1 \tag{5.19}$$

Where N_x and N_y are the x and y coordinates of N.

We already have the coordinates of points N and E. We just need to find the coordinates of Q.
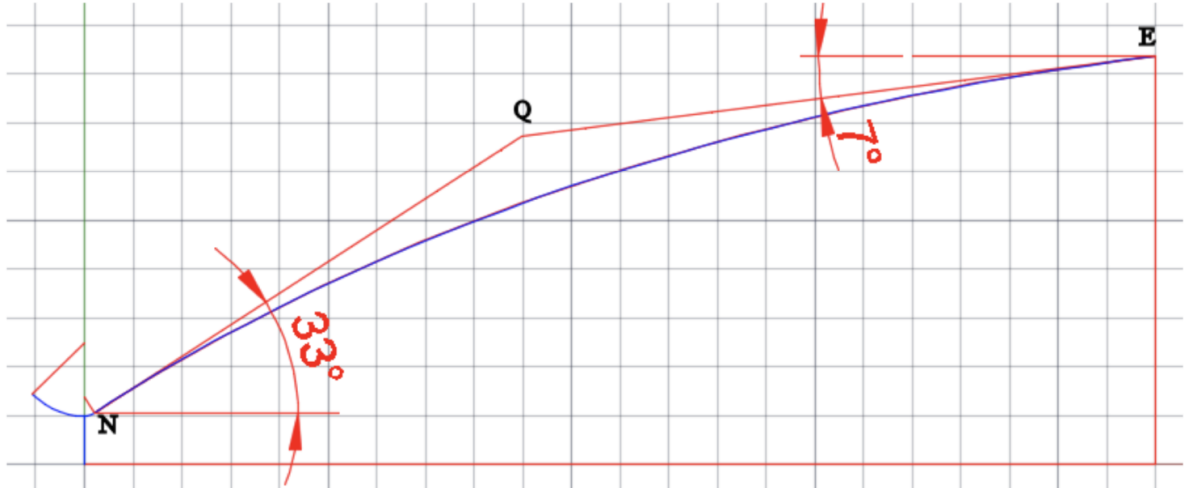
Figure 5.11: Finding Point Q from http://www.aspirespace.org.uk

Point Q is the intersection point of a line extending from point N with angle θ_n with respect to the horizontal and the line extending from point E with angle θ_e with respect to the horizontal.

A Matlab script can be written based on this algorithm that takes the throat radius and expansion ratio as inputs and outputs the coordinates of the nozzle profile.

5.4 Nozzle Thermal Considerations

One of the greatest challenges with nozzles especially with fuel like Ammonium Perchlorate Composite or Ethalox, is thermal management. These fuels have very high combustion temperatures (around 3000K), and the melting point of steel is around 1700K highest. As a result, we need a cooling solution. Two options are regenerative cooling or ablative cooling.

Regenerative cooling involves running fuel through a jacket around the nozzle in order to draw heat from the nozzle into the fuel. This is highly impractical due to its complexity (machining regenerative channels is very difficult).

This leaves the option of ablative cooling. The idea of ablative cooling is that instead of the heat being absorbed into the structure of the nozzle and melting it, the heat goes into some material, which is then ejected from the nozzle leaving fresh ablative material underneath it.

In practice, this would involve having a nozzle made out of graphite, or having a graphite insert in a steel nozzle carrier. As the hot gases flow through the nozzle, the graphite is slowly eroded away, taking the heat with it.