Abstract

We investigated the development of a Liquid Oxygen (LOX)/Liquid Methane (LCH4) engine that could be used a second stage for a NanoSatellite specific launch vehicle taking advantage of additive manufacturing. After an initial rocket sizing, we focused on the development of the injector and igniter in order to make tangible progress as a small team in a three month time period. We chose a pintle injector because of its resistance to combustion instabilities and manufacturing simplicity. Over the quarter, we investigated various designs and began water flowing them. Simultaneously, we chose a spark torch igniter for its reliability, even ignition, and ability to be tested and used independently of the main engine. We focused on creating and improving the igniter design through a literature review, analysis, CFD simulations, and feedback from industry engineers. Additionally, we began laying the foundation for a complete engine design and heat transfer analysis for regenerative cooling in the main chamber. Our process, progress, and lessons learned are discussed below.

Table of Contents

Abstract
Table of Contents
Introduction & Background
  General Sizing
  Expert Feedback
  Our Approach
Injector Design & Theory
  Injector Background
  Injector Selection
  Key Design Parameters of a Pintle
  Pintle Design
Injector Manufacturing & Testing
  Additive Manufacturing
  Waterflow Testing
Igniter Theory/Analysis
  Igniter type options
  Important Parameters for Spark Torch
  Feedback on design and analysis
Igniter CFD
  Modeling Advice
  Mesh Generation
  Fluent Model
  Cold Flow Results
Igniter Design & Manufacturing
  Design Iterations
Manufacturing
Test Plan
Thrust Chamber Assembly
  Chamber and Channel Design
  Heat Transfer Analysis
  Regen for LOX/Methane Specifically
Conclusions and Next Steps
References
Appendix 1: Rocket Sizing code
  Algorithm walk - through
  Source Code
Introduction & Background

Liquid rocket engines are appealing because they have higher performance than either hybrids or solids and are throttleable. This is critical for orbital insertion, which is why liquid rocket engines are almost always the propulsion of choice for upper rocket stages. Additionally, with the explosion in applications for small satellites, the desire for NanoSat specific launch vehicles has continuously increased. For these reasons, the goal of this project was to investigate a Liquid Oxygen (LOX)/Liquid Methane (LCH4) engine that could be used a second stage for a NanoSatellite specific launch vehicle taking advantage of additive manufacturing.

There are several reasons that the LOX/LCH4 propellant combination is appealing. One, methane could potentially be manufactured on Mars using the Sabatier reaction, a process that produces methane from the reaction of hydrogen and carbon dioxide at very high temperatures. Second, methane has a higher density than hydrogen, requiring less tank volume and less piping. In addition, it is easier to store because passive cooling is enough to keep it cryogenic since Methane is liquid at around 110 K compared to Hydrogen’s 20 K. Third, methane is very cheap and has quite high performance. For these reasons, many companies have been working to develop LOX/LCH4 engines for large scale engines, most notably SpaceX and Blue Origin.

To meet the size requirements for a NanoSat launcher, it is required to use a regeneratively cooled chamber - carrying the weight necessary for an ablative system is unacceptable given the extended burn times. However, as far as we know, no one has ever created a regeneratively cooled LOX/LCH4 engine at the scale necessary for a NanoSat specific launch vehicle. This was the focus of our program.

Building off a common thread for small engine development, we planned to take advantage of Additive Manufacturing. AM would allow us to develop intricate internal geometries necessary for low mass and cooling channels that otherwise would be impossible, especially without access to a complicated machining infrastructure. Many groups from Ventions to NASA Marshall have shown that additively manufactured parts can withstand the heat and force of a rocket engine, and while there are hurdles that remain, it is quite promising.

General Sizing

We began sizing a 1000 lbs class oxygen/methane pressure-fed rocket engine for the second stage of our sounding rocket. Aside from propellant tanks, the rocket would carry an onboard tank of helium for propellant pressurization, an avionics module, parachute, and small payload. To help characterize the requirements for the engine from a systems standpoint, we put together a matlab script (detailed in the appendix) to size the rocket. The code interfaces with CEA to calculate combustion characteristics, and uses basic sizing equations described in appendix for the rest of its calculations. Given relevant parameters including engine OF ratio
(mass ratio of oxidizer to fuel), target mass, and desired thrust level, the code estimates delta-V for the rocket, as well as burn time, a mass breakdown of all major components of the rocket, and total length.

Many trade studies were completed to set requirements for the full rocket system, but one of the first and most important results regarding the engine to come from this study was the necessity of liquid propellants. From a thermodynamics standpoint, the lower enthalpy of the propellants in liquid form makes little difference compared to the heat of reaction, and CEA calculations show that combustion temperature and specific impulse for gas phase propellants is only slightly higher than that of liquid phase. From a systems standpoint, the much higher density of liquid propellants allows for much more mass to be stored in a tank for a given volume and storage pressure, greatly reducing tank weight and making the mission feasible.

Next a trade study was run to determine the ideal oxidizer to fuel ratio for maximizing the specific impulse (Isp) of the engine. Specific impulse is a normalized ratio of thrust to mass flow, and is used to measure the performance of the rocket. Higher Isp allows a rocket to gain more delta V with the same propellant mass. The trade study showed the maximum Isp occurs at an OF ratio of 3.
Finally we ran a trade study to see the effects of chamber pressure on vehicle performance. For each chamber pressure, nozzle aspect ratios were chosen between 2 and 8 to maximize delta-V. The plot shows that lower chamber pressures yield a larger delta-V than higher pressures. This can be explained by the reduced mass of lower pressure propellant tanks and pressurant tank freeing up room for extra propellant mass, and allowing for a longer burn. However, the increased propellant mass makes the rocket longer, so a chamber pressure of 250 psi was chosen as a compromise.
Expert Feedback

With an idea of what we were aiming for, we set out to chat with people who had experience with liquid rockets and LOX/LCH4 engines. Notably we talked to Adam London CEO of Ventions, Jeff Thornburg head of Raptor at SpaceX, and several graduate students at Purdue who worked on their LOX/LCH4 development project. Here are the important lessons from each conversation for our overall approach:

Adam London

● While originally we were aiming for 1000 lbs of thrust for our first engine, Adam London quickly suggested that we drop down to 250 lbs in order to lower cost for the engine and testing and simplify some of the issues. He said that from their experience, there is tremendous difference between 250 and 500 and 500 and 1000 lbs and starting at 1000 lbs was not trivial.

● We had hoped to be able to find a test stand that we could use, rather than make one at Stanford. However, Adam pointed out that few people will have dual cryo test stands. Adam also thought it would take a decent amount of work to adjust someone else’s test stand for our efforts. As we feared, Adam also didn’t expect that most companies would want to deal with the liability issues.

● Adam also suggested that we run our igniter off of a separate GOX/GCH4 source to begin in order to decouple some of the problems we would face.

● We had thought about making a mobile platform that we tested by driving out to the desert, but while this would be possible, Adam suggested that it would not be conducive to a real development project since testing frequently is critical.

Purdue Students

Several generations of graduate students at Purdue worked extensively on LOX/methane thrust chamber assemblies for a lunar lander prototype vehicle being developed by NASA. The students were given initial requirements from NASA for the thrust range, specific impulse, run duration, and maximum tank pressure. Their reports can be found here. The main takeaways from their work were:

● They decided regen with Methane was too difficult because it tends to boil in the cooling channels. They turned to film cooling, but even so they had to supercharge their Methane to get the pressure high enough for a pressure fed system. To avoid the cost of Helium, they used gaseous methane to pressurize the liquid.

● They also chose a pintle injector due to its simplicity and manufacturing ease, but had tremendous difficulty achieving uniform flow through the annular gap. In their design,
flow was entering their internal manifold through only one inlet. After a few waterflow tests with different manifold designs, they were able to find one that provided uniform flow through the annular gap, but it took significant testing.

- Another challenge they faced for the injector was whether to make the pintle LOX or Fuel centered. Based on CFD analysis, they decided to make the pintle LOX-centered to avoid having large amounts of oxygen close to the wall.

- Arguably the biggest difficulty according to the Purdue team was finding a suitable source of liquid methane, since liquid methane is typically only sold in very large quantities and the CO2 in natural gas makes it imperfect. They spent a semester building a test setup to manufacture liquid methane by cooling methane gas with liquid nitrogen, which gave them enough methane for short tests.

- In order to focus on a few problems at a time and validate their analysis, the Purdue team began with a thick walled development chamber with thermal couples everywhere, instead of going straight to an actively cooled chamber. This allowed them to get testing quickly and focus on the injector and igniter design.

Jeff Thornburg

- Jeff stressed that SpaceX was very interested in work on regeneratively cooled LOX/Methane engines since not many people have looked into this. Despite the difficulties, Jeff was confident that regen for a 100/200 lb engine using methane could be done.

- Jeff argued that we should start with a general energy balance to scope our engine and then focus on designing each individual piece.

- Jeff agreed that make a development engine without regen was a good place to start. He suggested also testing the regen by using it to cool the engine while it is running, but having it be on a separate loop that did not go through the injector into the chamber. He also pointed out that the injector is very sensitive to the temperature of the methane, and therefore it might not be possible to use the same-injector for a non-regen and a regen chamber.

- For instabilities, Jeff thought we would be fine to start doing hand calculations on the longitudinal acoustic modes based on the geometry of the chamber. He also reminded us that chugging and feed system coupled instabilities could be an issue.

- For the pintle injector, Jeff agreed that doing a LOX centered design would likely be better to avoid the oxygen rich recirculation zones as Purdue had described. He also said that the most important characteristic was the momentum flux ratio. He agreed that water flows would be the best first step to characterize the general pattern and the Cd of the various holes.
Another student group we found at California State University developed a 1000 lbf LOX/methane rocket engine for a prototype launch vehicle in 2008. They were guided by Garvey Spacecraft Corporation. The engine used a self-impinging injector with an unlike triplet pattern and an ablative chamber with film cooling. The students found the unlike triplet pattern provided overall good mixing and atomization. One of the biggest challenges the group had was finding a source of methane. In the end, they used liquefied natural gas (LNG) for their tests. They had a successful flight test with a burn time of 5-6 seconds and the vehicle reached an altitude of approximately 5,500 feet.

Additionally, Armadillo Aerospace successfully tested a 4000 lbf engine with a pintle injector with no combustion instabilities.

**Our Approach**

Based on our initial sizing and the feedback we received from various people, here is how we decided to proceed focusing on a 200 lb engine with an OF ratio of 3 and a chamber pressure of 250 psi, which required a oxidizer mass flow rate of .625 lb/s and a fuel mass flow rate of .21 lb/s. We used this information to focus our efforts on designing a pintle injector and a spark torch igniter, as well as to begin the design of our combustion chamber as described below.
Injector Design & Theory

Injector Background

In a liquid rocket engine, the injector is responsible for introducing the propellants into the combustion chamber in a way that causes the propellant streams to atomize, vaporize, and mix uniformly with the correct oxidizer to fuel ratio. There are many different types of injector designs.

The design of an injector is influenced by many factors including propellant combination, state of propellant (e.g. liquid, gas, gel), cooling method for the combustion chamber (e.g. uncooled, ablative, regeneratively cooled), operating conditions (including chamber pressure, O/F ratio, chamber temperatures), transient conditions, chamber geometry, and throttling requirements.

Since the early stages of rocketry, countless types of injectors have been designed, built, and tested. The most common injector types are the impinging-stream injector, shear coaxial injector, and pintle injector. With impinging-stream injectors, propellants are injected through a number of separate small holes in a way that either fuel and oxidizer streams impinge on each other (unlike pattern) or in a way that fuel impinges on fuel and oxidizer impinges on oxidizer (like-on-like, self-impinging). In the self-impinging case, the two fuel streams form a fan which breaks up into droplets (shown in the third image of Figure 1).

**Figure 1:** Examples of Impinging Injectors

A shear coaxial injector is composed of two concentric cylinders. It has an inner post from which one propellant emerges, while the other propellant flows in the tiny annular gap around between the inner post and the outer cylinder. The two flows are injected at differing velocities, with the shearing between them providing the mechanism for mixing and dispersal. Both flows are injected normal to the injector face, as opposed to at an angle as in the impinging injectors.
A pintle injector has an inner post, called a pintle, with an annular gap around the pintle like that of a coaxial injector. The pintle is shaped like a capped cylinder, with a series of holes or slots near the end which ejects the fuel radially. This fuel collides with the annular sheet of oxidizer around the base of the pintle, creating vigorous mixing and atomization.

Injector Selection

For our project, we narrowed our choices to a self-impinging injector, a shear coaxial injector, and a pintle injector. The self-impinging injector has the advantages of good mixing, being well studied, and having less combustion instability issues than an unlike impinging injector. However, precise alignment and tight tolerances are required with the orifice holes, making it difficult to fabricate. In addition, these type of injectors have had numerous combustion instability issues with LOX/LCH4, and would require stability devices such as baffles or acoustic cavities. The shear coaxial injector also has good mixing and the orifice holes are normal to the surface, however, the shear coaxial injector has also had combustion instability issues with LOX/LCH4 and again would probably require stability devices.
The pintle injector, on the other hand, has not had any instances of combustion instability, eliminating the need for stability devices. It also has far fewer injection sites than a shear coaxial injector a self-impinging injector, and is simpler to manufacture, reducing the cost.

One of the main reasons pintle injectors don’t have a history of combustion instabilities is because the momentum from the spray fan creates two recirculation zones, as show in the image below.

Figure 4: Recirculation zones in a pintle

Since the combustion process is curved, the energy release is not on a planar surface, and is away from the pressure antinodes of the chamber.

Despite these benefits, there are some key challenges with the pintle. First, the pintle tip lies in a recirculation zone and therefore is subject to high heat fluxes and can potentially break off. Often times, rounded pintle tips are used, and they're designed to be removable so they can be replaced for testing or in case of damage. Second, it is difficult to achieve uniform annular flow around the post, in part because the annular gap is difficult to machine under such tight tolerances.

Despite these challenges, we have selected to use a pintle injector in our design for the reasons listed above and the fact that a pintle injector has successfully worked with the propellant combination of liquid LOX/CH4 as discussed previously.

**Key Design Parameters of a Pintle**

One of the key design parameters of a pintle is the total momentum ratio, defined as the ratio of the radial momentum to the axial momentum. This ratio needs to be properly designed in order to create a spray that will provide vigorous mixing and atomization, create low temperature recirculation zones near the mantle, and not carry the oxidizer gas near the chamber walls.

Below is an image and table that has a list of other key design parameters.
**Table:**

<table>
<thead>
<tr>
<th>Key Design Parameters</th>
<th>Definition</th>
<th>Typical Values</th>
<th>Reasoning</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Total Momentum Ratio (TMR)</strong></td>
<td>Ratio of radial to axial stream momentum</td>
<td>~0.5 to 1.5</td>
<td>Close to 1 has been proven optimal from design experience. As TMR increases, the fan angle increases. If ratio is too low, fan angle is extremely small and the spray will be very close to tip. If ratio is too high, fan angle is high and the flow will impinge directly on chamber walls. This value is closely coupled with whether the pintle is oxidizer or fuel centered. If the pintle is oxidizer centered, a TMR greater than 1 will lead to oxidizer impinging directly on the chamber walls, possibly creating damage and hot spots. So, typically, the TMR will be between 0.5 and 1.0 in this case.</td>
</tr>
<tr>
<td>Oxidizer/Fuel Centered</td>
<td>Propellant that flows down the inside of the pintle and shoots out radially</td>
<td>Large engines typically are oxidizer centered, small engines typically are fuel centered.</td>
<td>Smaller engines are typically fuel centered because otherwise, the annular gap size becomes very small (tolerances &lt; 0.0001 in) Larger engines are typically oxidizer centered because the annular flow</td>
</tr>
</tbody>
</table>

---

**Figure 5:** Pintle Parameters
makes up the majority of the gas in the recirculation zone and if this was the oxidizer, it could move along the chamber walls and cause damage.

<table>
<thead>
<tr>
<th>Blockage Factor</th>
<th>Ratio of total hole/slot circumferential length to circumference of the pintle</th>
<th>~0.5</th>
<th>Number of holes/slots in the pintle tip is usually around 20 – 36</th>
<th>A set of secondary holes is often placed just downstream of the primary holes (as shown in image). These holes are typically smaller in area.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ratio of Chamber to Pintle Diameter</td>
<td>~3 to 5</td>
<td>There needs to be enough space for the annular flow and radial flow to mix and atomize for the combustion. There will probably be structural issues if the pintle becomes too skinny.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Skip Distance ($L_s$)</td>
<td>Length that the annular flow must travel before impacting the radial holes</td>
<td>Ratio of Skip Distance to Pintle Diameter is typically around 1</td>
<td>Large skip distances can decelerate the liquid because of friction against the pintle post (and this can change the momentum ratio). Short skip distances can lead to spray impingement towards the top of the combustion chamber.</td>
<td></td>
</tr>
</tbody>
</table>

There aren’t many academic studies on pintles or the physics of an annular sheet impinging on multiple radial jets. Pintles have mainly been designed from empirical methods. One of the main reasons they’ve been designed empirically is that there are only two main components to change and optimize for the annular and radial flows. These components are designed to be easily interchangeable which allows different configurations to be iterated through quickly.

**Pintle Design**

We determined the length of our skip length and pintle diameter from our chamber diameter and industry standards. We were able to roughly calculate the area of our annular gap and holes from the mass flow equation for an incompressible fluid through an orifice.

\[
m_{dot} = C_d A \sqrt{2 \rho \Delta p}
\]
Where \( C_d \) is the dimensionless discharge coefficient, \( A \) is the cross-sectional area of the orifice, \( \rho \) is the density of the fluid and \( \Delta P \) is the pressure drop. When the discharge coefficient equals 1, the injection velocity is a maximum for a given pressure drop. Smooth, well-rounded entrances to the injection holes lead to higher values of \( C_d \). The lowest \( C_d \) value is 0.61 for a sharp-edge orifice. The exact value of \( C_d \) cannot be determined until the injector part is manufactured and water flow tested. For our initial analysis, a \( C_d \) value of 0.61 was used.

The pressure drop across an injector is usually between 15 and 25% of the chamber pressure in order to obtain high injection velocities, which in turn leads to better atomization. A pressure drop of 20% was used for the calculations in the table.

<table>
<thead>
<tr>
<th>Total Momentum Ratio</th>
<th>0.79</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pintle Diameter</td>
<td>1 in</td>
</tr>
<tr>
<td>Skip Length</td>
<td>1 in</td>
</tr>
<tr>
<td># of Primary Pintle Holes</td>
<td>14</td>
</tr>
<tr>
<td># of Secondary Pintle Holes</td>
<td>14</td>
</tr>
</tbody>
</table>

**Fuel Centered**

<table>
<thead>
<tr>
<th>Primary Hole Diameter</th>
<th>0.023 in</th>
</tr>
</thead>
<tbody>
<tr>
<td>Secondary Hole Diameter</td>
<td>0.0162 in</td>
</tr>
<tr>
<td>Annular Gap</td>
<td>0.0050 in</td>
</tr>
</tbody>
</table>

**Oxidizer Centered**

<table>
<thead>
<tr>
<th>Primary Hole Diameter</th>
<th>0.0310 in</th>
</tr>
</thead>
<tbody>
<tr>
<td>Secondary Hole Diameter</td>
<td>0.0219 in</td>
</tr>
<tr>
<td>Annular Gap</td>
<td>0.0021 in</td>
</tr>
</tbody>
</table>

**Flow Analysis**

Preliminary analysis was done using flow simulation in Solidworks of the flow inside the internal manifold and the post. The internal manifold was designed in order to have minimal pressure losses across it to increase the injection velocity of the fluid into the chamber and achieve an optimum total momentum ratio. We ran simulations on different geometries, and found a design that achieved a total momentum ratio of 0.79 with a LOX centered flow.
After talking with engineers from SpaceX, Blue Origin, and Armadillo Aerospace, we shifted our focus from running analytical simulations to water-flow testing with 3d-printed parts. They advised that running water flow tests on numerous geometries would be more efficient than running simulations on various geometries without validation of the model.

**Injector Manufacturing & Testing**

**Additive Manufacturing**

Early in the course of the project, additive manufacturing was decided as the manufacturing method for the injector assembly. This choice was primarily made so that the complex manifold geometry required between the regen channels and the pintle injector could be easily incorporated and manufactured. The first iterations of the manifold design would not need to interface with any regen channels since the initial chamber would simply be a thick-walled steel
body, but knowing the injector would be additively manufactured allowed for relative freedom in design without worrying about manufacturing capabilities. Using metal additive manufacturing for the final design also meant that plastic additive manufacturing could be easily employed for test printing parts and doing initial tests on fit, assembly, and waterflow.

The initial design of the injector included five separate, interchangeable pieces. This design was chosen such that the characteristic aspects of the injector would be modular, and each piece easily swapped out for a different design without having to remanufacture the entire injector. These five parts consisted of the pintle post, the holed ring at the end of the pintle post, the internal profile for the annular flow, a basic manifold, and a top connector piece to hold seal the top of the manifold. These parts are shown individually and assembled below.

![Figure 8: 1st Design Parts](image)

The initial manifold design was very basic and was not adequate for either testing or final manufacturing of the overall assembly. A second version of the manifold part was designed such that it could properly interface with a chamber that was regeneratively cooled. This design was more advanced than what we intended to begin testing with but it was used to get a quote for metal printing of the assembly to help evaluate budget and testing approach. It was determined that final manufacturing of the full injector assembly would be on the order of $2,500. This meant that it would not be practical to manufacture multiple designs in metal and helped drive the decision to test in plastic as much as possible.
Waterflow Testing

Design 1

To validate preliminary designs of an injector, waterflow testing is typically done to validate the expected flow patterns, to calculate the discharge coefficient of the orifices, and to measure the spray fan angle.

Below is a figure showing a waterflow test of a pintle injector. The first image depicts the annular flow, a uniform sheet around the pintle. The second image shows the radial flow shooting out of holes or slots near the pintle tip, and the third image shows the two flows combined, forming a fan spray.

Figure 10: Waterflow testing of a pintle injector

This is a typical procedure for waterflow testing a pintle injector:

Table: Waterflow test procedure
<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Place test article over a bucket.</td>
</tr>
<tr>
<td>2</td>
<td>Connect hose to the center flow inlet of the injector.</td>
</tr>
<tr>
<td>3</td>
<td>Start water flow and adjust flow rate until input pressure is 60 psi.</td>
</tr>
<tr>
<td>4</td>
<td>Verify flow shoots out radially from all the primary and secondary holes.</td>
</tr>
<tr>
<td>5</td>
<td>Collect water for 1 minute.</td>
</tr>
<tr>
<td>6</td>
<td>Weigh water using a scale.</td>
</tr>
<tr>
<td>7</td>
<td>Repeat steps 2 to 6 as necessary.</td>
</tr>
<tr>
<td>8</td>
<td>Rinse out bucket and connect hose to the annular flow inlet of the injector.</td>
</tr>
<tr>
<td>9</td>
<td>Repeat steps 3 to 7.</td>
</tr>
<tr>
<td>10</td>
<td>Using these measurements, the Cd can be calculated for the average orifice using the mass flow equation for an incompressible fluid through an orifice.</td>
</tr>
<tr>
<td>11</td>
<td>Connect water to both oxidizer and fuel inlets.</td>
</tr>
<tr>
<td>12</td>
<td>Visually examine the flow and measure the spray fan angle.</td>
</tr>
</tbody>
</table>

Prior to testing, another redesign of the manifold was done to properly interface with a single water line in order to perform the first workflow test. In designing the manifold for this test, the five separate parts of the assembly were forced to be merged into one single part. This version of the design was not ideal as it would require us to print a new version to change any one component, but the manifold design employed made assembling or changing various pieces of the design difficult if not impossible. With only one connection point and the need for both a center and annular flow, the manifold needed to reach outward from the center post to the outer edge of the lower annular manifold. This can be seen below in Figure 11. In addition, three separate inlet pieces were designed so that testing could be done for only the annular flow, only the center flow, and both flows simultaneously.
Multiple issues were encountered when the first waterflow test was attempted with this design. The first issue was with the three inlet pieces for the three flow cases. Tolerancing on the diameter and length had not been calculated correctly, and the parts proved to be very difficult to insert or remove from the rest of the injector. The rough surfaces from the printing process magnified the tolerancing issue. More concerning was that the water did not appear to flow through the annular manifold as intended, very little if any annular flow was noticeable during the test. The problem here was likely two fold. First, removing the printed support material from the manifold passageways proved to be difficult, and created partial blockage. In addition, the walls of the manifold were thin and allowed noticeable leakage during testing.

From this first test, it was clear that several changes in the design would be necessary, and that water flowing successfully with plastic printed parts would likely be more challenging than anticipated. On the plus side, the center flow during the test did appear to work as desired.

For the next waterflow test, the design was moved back to multiple parts so that making changes to individual pieces would be possible. In order to do this, the injector would now need
to have two separate flow inlets. This was achieved by placing a tee on the single water line from before. The multiple parts were designed to be held together by several bolts and a couple of press fit pins. No sealing parts or surfaces were included as the hope was that leakage would not be significant under relatively low pressures (60 psi).

![Figure 13: Waterflow #2 manifold and assembly design](image)

Additionally, the pintle tip was designed to be removable so that we could swap it quickly and test different sized holes for the radial flow.

Leakage once again proved to be an issue, even with this new design. For the annular flow, the combination of the bolt pattern and press fit pins did not provide adequate sealing, and led to significant leaking on both the top and bottom of the assembly. We also noticed that annular gap between the post and manifold was not consistent. We concluded that plastic printing was likely not capable of the tolerance needed for that gap, and that post machining might be the best solution in order to achieve that tolerance.

Leakage was also an issue for the center flow. The pintle tip was designed to thread onto the post for easy iteration. The two parts did thread together as intended, but did not provide adequate sealing.

We also noticed that there continued to be leaks straight through the plastic walls in all parts. This from the printing process, which extrudes plastic to form a grid, leaving small air gaps and making the finished parts slightly porous.

For both the center and the annular flow, there was too much leakage to obtain any useful data. However, we could use the manufacturing lessons to revise our design.

We determined that the issue with our connection between the pintle tip and post was that threading the tip inside of the post created an open flow path out the edges of the sealing surface. To prevent this problem, we redesigned to have the pintle tip thread on the outside of the post.

Our next attempt was to print the two parts as two solid cylinders and then post machine them with metal taps and dies to create the correct threads. The challenge became clamping the
parts tight enough to remain in place while machining, without deforming the part. We observed that the required tool pressure to cut the threads tended to over stress the plastic and break the part. Several iterations in part design, tolerances, and threading technique were needed to create a process that gave fair results. The remaining challenge was in getting both threads straight such that the outer diameter of both parts would line up for a smooth annular flow surface. It is believed that with careful fixturing this could be achieved.

![Figure 14: Sample printed and threaded post parts](image)

**Water proofing through Acetone treatment**

Research into sealing printed plastic parts revealed two methods: acetone and epoxy. In the case of using epoxy, the parts would be sealed by coating sealing surfaces with a layer of epoxy. For our design, coating all necessary surfaces did not seem practical. Maintaining surface smoothness was also a concern and epoxy did not seem like the best option for that either.

In the case of acetone, parts are briefly exposed to either acetone liquid or vapor, and the plastic melts enough to seal the porous air gaps. Tests were done first with liquid acetone and then acetone vapor. In both tests, part deformation was an area of concern. Leaving a part in liquid acetone meant much faster melting and deformation of the part. The concern was that getting the parts in and out quick enough while still getting an even coating would be a problem for liquid acetone. Acetone vapor was then tried as a preferred alternative.

Producing acetone vapor in a safe manner and securing the part became the primary problems to address. The acetone was boiled using a hot plate and the part was held within a closed volume over a boiling acetone bath. The acetone was contained within a standard cooking pot over a hot plate. The part was suspended from the top of another glass bowl placed upside down in the bath using a string and tape. The tendency of the tape was to peel off and release the part within about five minutes. The bowl and pot were also too close in diameter which made removal of the smaller bowl difficult.
Testing revealed that acceptable exposure time for acetone vapor was likely less than two minutes. The parts that had been left in for five minutes were too warped to be used, even if they did seal better than before. The part tested for only two minutes still showed some signs of warping and so a shorter duration would be needed.

The setup used for these tests was not ideal in trying to secure the part or easily remove the part from the setup. It is believed that this method with acetone vapor would be the best option for trying to seal plastic printed parts in the future. While we did see that the parts sealed better when exposed to low pressures (~10psi), it is still unclear if the parts would seal under higher pressures.

Alternatively, printing using a printer with higher resolution, a different printing method, or using a different material might provide non-porous parts more effectively.
Igniter Theory/Analysis

Igniter type options

Pyrotechnic

The most basic and most common type of igniter is a pyrotechnic igniter. It consists of a small grain of solid propellant ignited by an electric current. The heated stream of exhaust products from the reaction enters the main chamber and provides ignition energy to the main propellants. Pyrotechnics are cheap and relatively easy to manufacture or buy, but can only be used once. When designed and implemented properly, they can be very reliable, but they require additional safety precautions.

![Figure 16: Example of pyrotechnic](image)

Resistance heaters

Resistance heaters work by driving a large current through a small element, such as a glow plug used in diesel engines, and heats it red hot. The element protrudes into the main chamber and heats the propellants until they ignite. They are very sensitive to the flow patterns around the heating element and the chamber pressure before ignition. Insufficient flow around the heating element tip due to wall proximity prevents the heat from convecting to the propellants, and dumps all the heat to the wall. Too much convection cools the tip without heating any volume of propellant to the point of ignition.

Testing of a glow plug ignition system at NASA Glenn Research Center demonstrated this sensitivity as there was a very specific chamber pressure range in which they could get consistent ignition. They determined that the minimum glow plug temperature for igniting oxygen and methane was 1230 K, and additionally required an ox to fuel ratio between 9 and 14 (very high compared to typical OFs for main chamber combustion). The glow plug took 7 seconds to warm up to that temperature, and drew 83 W while operating.
Resistance heater igniters are reusable as long as the element is not damaged by heat or chemical action from the combustion process, but they are not too reliable. The NASA researchers noted in their paper that they broke many glow plugs during the testing process.

**Spark**

Spark igniters work by creating a large voltage between two closely spaced metal tips. When the voltage exceeds the breakdown voltage, it ionizes the gas between the two tips and arcs across, creating a spark. Spark plugs are often used in internal combustion engines, and can be used to ignite liquid or gaseous propellants. The breakdown voltage is dictated by Paschen’s Law, and is dependent on the pressure and composition of the gas between the tips as well as their spacing.

Researchers at NASA also experimented with spark ignition. They found that the sparks would reliably ignite with spark energies above 70 mJ and spark rates of 150 sparks per second, requiring only 10 W to run. Additionally, they found that often times the first spark was sufficient to produce ignition.

One drawback of spark igniters is the ignition site produced by the spark is very small, and must spread in order to establish combustion throughout the chamber. Depending on the size and geometry of the chamber, this uneven startup can cause an undesirable transient and damage the combustion chamber. Multiple spark plugs are sometimes required to ensure more even ignition.

![Spark Plug Internals](image)

**Figure 17: Spark plug internals**

**Augmented Spark (Spark Torch)**

Augmented Spark Igniters, often called spark torch or torch-style igniters, are an extension of spark ignition concepts. Spark torches igniters consist of a side chamber connected to the main combustion chamber with a spark plug and a small propellant feed controlled independently of
the main chamber. This allows for good control of the conditions around the spark plug, and leads to more reliable ignition.

By performing the spark ignition using a pilot flow of propellant, spark torch igniters solve one of the big problems with plain spark and resistance heating ignition systems, namely that there is no way to verify successful ignition with correct timing without opening main propellant valves. Improper ignition could lead to the accumulation of unburnt propellants in the combustion chamber, which can cause a hard start or explosion. The flame from the spark torch igniter provides a large and diffuse ignition source for the main propellants ensuring even ignition.

The spark torch’s even ignition, reusability, reliability, and capacity for separate ignition verification and testing made it the ideal candidate for an igniter, with pyrotechnic ignition as a second choice. Therefore, we chose to investigate a spark torch for our set up.

![Figure 18: Augmented spark igniter, modified to work with glow plug](image)

**Figure 18: Augmented spark igniter, modified to work with glow plug**

### Important Parameters for Spark Torch

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Val</th>
<th>Description</th>
<th>Rationale for choice</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass Flow Rate</td>
<td>3 g/s</td>
<td>Mass flow rate to the torch igniter was chosen to be 2% of the main propellant mass flow.</td>
<td>This provides a large ignition flame to ensure ignition occurs as evenly as possible. The torch will be outputting a volumetric flow rate approximately 50 times greater than the liquid propellant flow from the main injector prior to ignition, filling the chamber with hot fuel rich gases.</td>
</tr>
</tbody>
</table>
The torch will run fuel rich compared to the main combustor to spare the igniter body (see Torch Cavity Temperature). Spark ignition favors ox-rich mixture in 8-12 range (from research and industry experts), too hot for the torch body. The breakdown voltage of methane is approximately 23% less than that of oxygen, making it easier to spark across a more ox-rich mixture. NASA GRC researchers chose similar bulk OF of 2 for those reasons, but maintained an OF of 18 at spark plug by injecting most of the fuel after the ignition site in the form of film cooling. Chosen as a tradeoff between melting the torch inner surface while having enough temperature and thermal energy to effectively ignite the main propellants. The igniter does not have the regenerative cooling like the chamber, so lower OFs lead to lower temperatures, keep it from melting for short duration (~1s) firings. Around 1.6 is the fuel rich boundary for ignition, so OF must be above that. Also want to ensure that flame sheet at the exit has a high enough theoretical temperature that with heat losses the real flame will still have enough temperature to ignite the propellants.

<table>
<thead>
<tr>
<th>OF Ratio</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chosen as a tradeoff between melting the torch inner surface while having enough temperature and thermal energy to effectively ignite the main propellants. The igniter does not have the regenerative cooling like the chamber, so lower OFs lead to lower temperatures, keep it from melting for short duration (~1s) firings. Around 1.6 is the fuel rich boundary for ignition, so OF must be above that. Also want to ensure that flame sheet at the exit has a high enough theoretical temperature that with heat losses the real flame will still have enough temperature to ignite the propellants.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>GOX Density at ignition</th>
<th>1.49 kg/m$^3$</th>
</tr>
</thead>
<tbody>
<tr>
<td>During cold-flow, propellants do not have enough specific density to raise the cavity pressure above ambient. The propellants are assumed to be stored in a full K-bottle (~2000 psi) at room temperature. Expansion is modeled as a real gas throttling process, based on the assumption that the gas will choke and lose most of its pressure across a single component. The gas would do very little work, which combined with the small heat transfer from outside would average to an isenthalpic process. The final state of the propellants is 14.7 psi for both, with GOX temperature of 263 K and CH4 temperature of 238 K. Propellant density is not as easily controllable, but it does affect the operation and performance of the spark torch. Variations in density change the momentum flux ratio and therefore the mixing behavior.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>CH4 Density at ignition</th>
<th>0.83 kg/m$^3$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spark energy depends on breakdown voltage of mixture in spark gap. Spark energy creates a small region of high temperature ionized gas and provides the activation energy for the initial combustion reactions. Numerous NASA papers point to minimum spark energy needed for ignition to be 60 – 70 mJ, with spark rates of 150 – 200 sparks/s. Higher spark energies provided slightly more reliable ignition. For a better treatment of spark ignition mechanisms, see the ME 372 Course Reader. Chosen values were within or exceeding minimums gleaned from references. NASA papers showed the first spark often ignited the torch, but if it does not and the spark rate is too low, unburnt propellants can accumulate in the cavity and cause overpressure during ignition. Upper limits on both are imposed by hardware limitations and the desire to minimize igniter power requirements for an eventual flight system. During testing, they will be varied to validate predictions.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Spark Energy</th>
<th>90 mJ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spark energy depends on breakdown voltage of mixture in spark gap. Spark energy creates a small region of high temperature ionized gas and provides the activation energy for the initial combustion reactions. Numerous NASA papers point to minimum spark energy needed for ignition to be 60 – 70 mJ, with spark rates of 150 – 200 sparks/s. Higher spark energies provided slightly more reliable ignition. For a better treatment of spark ignition mechanisms, see the ME 372 Course Reader. Chosen values were within or exceeding minimums gleaned from references. NASA papers showed the first spark often ignited the torch, but if it does not and the spark rate is too low, unburnt propellants can accumulate in the cavity and cause overpressure during ignition. Upper limits on both are imposed by hardware limitations and the desire to minimize igniter power requirements for an eventual flight system. During testing, they will be varied to validate predictions.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Spark Rate</th>
<th>150 spark/s</th>
</tr>
</thead>
</table>
Working pressure

<table>
<thead>
<tr>
<th>Working pressure</th>
<th>275 psi</th>
<th>Pressure inside the torch body when both torch and chamber are in operation. Exit diameter of torch was narrowed to .18” to achieve this pressure, the diameter calculated by solving mass balance and pressure balance across the exit using CEA combustion properties.</th>
<th>Chosen to be 10% higher than chamber pressure to ensure that normal fluctuations in chamber pressure (defined as &lt;10%) will not be able to push combustion gases back into the torch. Higher working pressures would lead to higher torch only pressures, which increases heat flux to the torch walls and shortens the maximum allowable continuous firing time for the igniter before internal melting. From a safety standpoint higher pressures are potentially more dangerous, and unneeded here.</th>
</tr>
</thead>
</table>

Torch Cavity Temperature

<table>
<thead>
<tr>
<th>Torch Cavity Temperature</th>
<th>2925 K</th>
<th>Determined using CEA according to the OF ratio in the torch, 400 K lower than the main combustion chamber temperature.</th>
<th>Driven by OF ratio.</th>
</tr>
</thead>
</table>

Torch Cavity Diameter

<table>
<thead>
<tr>
<th>Torch Cavity Diameter</th>
<th>0.5 in</th>
<th>Want to have a smaller torch cavity ensures that propellant cannot collect excessively prior to ignition and cause overpressurization. Maintaining surface area to volume ratio low ensures that large thermal mass of torch body does not quench flame.</th>
<th>Torch cavity volume calculated using approximate $L^<em>$ value of 35, where $L^</em>$ is the ratio of chamber volume to exit area for a combustor. $L^*$ is proportional to residence time of gases and is approximately constant for a given fuel combination.</th>
</tr>
</thead>
</table>

Momentum Flux Ratio

<table>
<thead>
<tr>
<th>Momentum Flux Ratio</th>
<th>27.5</th>
<th>Rough measure of jet penetration into the main flow, determines mixing behavior and deflection angle of the combined jet. Preliminary CFD shows that the oxygen jet stays concentrated and does not fully expand to occupy the whole chamber area, bringing the effective momentum flux ratio near 1</th>
<th>Having the actual momentum flux ratio close to 1 means where the jets intersect, the resultant stream should deflect at approximately 45°, which yields superior mixing compared to cases where the jet momentum fluxes are unmatched.</th>
</tr>
</thead>
</table>

Feedback on design and analysis

We discussed our design with Brian Evans, a postdoc from Dr. Cantwell's lab and engineer at Space Propulsion Group, who had previously designed and worked with methane oxygen spark torch igniters. He recommended a design with impinging gaseous propellant jets, noting the aerodynamic stability and good mixing of such a scheme combined with the simple design, compared to other schemes such as co-axial propellant injection. He recommended the torch igniter body material be brass or copper rather than stainless steel or aluminum, as their high thermal conductivity and maximum operation temperature conducts heat away from the inner surface faster, preventing melting and other damage due to excessive surface heating. In addition, brass and copper are more oxygen compatible than stainless or aluminum. He also recommended downsizing the mass flow on our initial design, and suggested we consider the heat output of the torch versus ignition requirements for the main chamber.
Jeff Thornburg from SpaceX agreed that a simple, dumb igniter was the way to start. From his experience, he thought the best plan was to bring the GOX in ahead of the electrode and carry the plasma from the spark into the impinging fuel stream to ensure combustion begins in an area with a proper mixture ratio. He also thought that bringing fuel in and mixing it around the circumference of the igniter was essential. For an actual test, he recommended keeping the igniter on the entire time or shutting it down and turning on a purge to prevent the chamber flow from entering the igniter cavity. With a spark torch, Jeff thought the biggest challenge would be getting repeatability in terms of mixture ratio where combustion begins.
Igniter CFD

The goal of the CFD analysis of the igniter was to gain an understanding of the flow and physical processes that were happening inside the igniter during its operation, and use these to influence and improve the design. Modeling combustion and the ignition process itself quickly showed to be a much larger and more complicated endeavor than we had time to allot, so we limited ourselves to modeling the cold flow in the igniter that would be seen prior to ignition, and draw our conclusions from those results.

Modeling Advice

Our first attempt at modeling the igniter in coldflow drew from minimal previous experience running CFD using Fluent, and was fraught with problems. It showed very poor convergence, crashed or grew unstable at times, and neglected to properly treat the turbulence near the walls. After the first attempt at modeling, advice was sought from different graduate students and post-docs in the ME department, namely Julia Ling, Riccardo Rossi, and Yee Chee See. They provided invaluable help for improving the CFD model. Firstly, they recommended we focus on improving the mesh. They recommended simplifying the geometry to remove extraneous features such as small fillets or bosses around the propellant inlets, which have little effect on the flow patterns, but greatly degrade the quality of the mesh around them. Noting the geometry was left-right symmetric, they recommended halving the region and using a symmetry plane, citing the flow’s similarity to the well studied problem of jets in crossflow to support that the flow itself would behave symmetrically in such a scenario.

![Comparison of basic mesh types](image)

Mesh quality is one of the key factors for obtaining numerical convergence of the residuals, high quality meshes with low skewness and reasonable aspect ratios will converge faster and more accurately. They also recommended the use of structured meshes wherever possible. Structured meshes provide much higher accuracy and faster convergence for a smaller number of elements than unstructured meshes. They recommended segmenting the model to isolate the prismatic sections for structured meshing, leaving unstructured meshes only on the complicated geometry.
In addition, they advised better treatment of the turbulence model. They recommended switching from standard to realizable k-epsilon, or considering k-omega SST model. They also advised and provided resources for ensuring proper y+ at the walls, and recommended using enhanced wall treatment, with y+ values close to 1. Lastly, they recommended an incremental approach to modeling, starting with simpler meshes, single species, and simple turbulence models and progressing step by step to the desired model, running each step to ensure convergence along the way.

No advice was given with regards to modeling compressibility or multi-species flow, both which are very large factors of the model. With additional time, we would have sought advice on modeling those factors, and cold-flow results would have been discussed with them to confirm the modeling approach.

**Mesh Generation**

The fluid volume was first split in half with a symmetry plane, and then split into six different regions. The intention was to separate the prismatic sections of the model, such as the two inlets, post injection mixing zone, and outlet from the rest of the model. These sections would be mapped using hexahedral elements in a structured manner with elements of 0.45 mm for the inlets and 0.35 mm for the mixing zone and outlet to create a better quality, higher accuracy mesh. The complicated region containing both oxygen and methane injection sites as well as the spark plug was meshed with a patch conforming tetrahedral scheme, with element sizes of 0.35 mm. After experimentation with swept and hexahedral meshes, a patch conforming tetrahedral scheme was chosen for the converging section of the outlet, noting that it created higher quality elements than the other candidates.

![Figure 20: Igniter fluid volume broken up into bodies](image)

**Inflation layer attempt**
Ideally, you would want the mesh to be as coarse as possible in all regions, while still fine enough to capture flow phenomena. Boundary layers along wall surfaces are the most common region where a finer mesh is desired, and a meshing control called an inflation layer. Inflation creates a set number of thin layers of mesh elements on the given surface. The layers can be the same height but often increase by a given scale factor as they get farther from the wall in order to prevent a sharp transition in element size between the last layer and the main mesh. The height of first layer is dependent on flow characteristics and the choice of turbulence model. This is touched upon in the modeling section, but for more explanation see the discussion in the ANSYS “Tips and Tricks” guide.

We attempted to add an inflation layer on the mesh described above, but were unsuccessful. After many failed attempts, a consultation of the ANSYS Meshing tool user guide showed that the inflation algorithms used by the default mesher cannot be used in hex dominant meshes, and only in certain circumstances for swept or multizone meshes. The nature of the geometry decomposition shown above suggested this should be possible to achieve, but we never successfully realized inflation layers on all the zones. This left us with the option of either using an unstructured mesh in the entire geometry, or switching to a different meshing tool. With limited time, we decided that it was not worth trying to learn a different tool, and the fully unstructured meshes required more elements and yielded slower convergence and choppier contours than the equivalent structured mesh. Thus, we decided to omit inflation layer from this preliminary run, and use mesh adaptation in Fluent to aid in the boundary layer capture.

![Image of mesh boundary](image.png)

**Figure 21: Unstructured to structured mesh boundary, showing smoother contours on structured mesh**

On a more general note, this inflation layer attempt highlighted one of the bigger lessons we learned about the ANSYS Meshing tool. Unlike previous ANSYS meshing tools like ICEM or Gambit, this tool is very automated. It is designed to produce good quality meshes, with little...
user input other than some high level sizing and other parameters. This makes it much faster and easier to create a mesh compared to other common tools, at the cost of losing control over the lower level mesh features. Certain meshes (including the boundary layer attempt above) are difficult, if not impossible to create with the tool, and at other times it creates a very poor quality mesh with highly skewed elements or throw errors in regions where one would not expect that mesh type to fail (and where other tools have succeeded). The ANSYS Meshing tool is very powerful and is a good first choice for mesh creation, but can be finicky at times and you cannot force it to make meshes it doesn't like.

Fluent Model
A k-epsilon turbulence model was chosen due to its robustness and simplicity. Results were first converged with the standard model, before switching to a realizable model. Standard wall functions were used. Viscous heating modeling was attempted, but due to the highly compressible nature of the flow, all attempted solutions using this option swiftly crashed.

A multi-species model was used to treat the mixture of methane and oxygen. The standard methane-air mixture was used, with the carbon dioxide and nitrogen excluded from the simulation. As this model treated cold-flow scenario, reactions were not modeled, just simple transport. This model required energy to be included in the simulation as well.

A symmetry condition was applied to the symmetry plane and an adiabatic wall condition was applied. A mass-inlet condition was applied at the oxygen and methane inlets, setting the inlet mass flow of 2 g/s for oxygen and 1 g/s for methane. Incoming oxygen was assigned a temperature of 270 K and methane 250 K, estimated from expanding oxygen from a high pressure gas cylinder to the test setup using real (non-ideal) gas properties. Inlet turbulence was set to a turbulence intensity of 7% and a hydraulic diameter of 4.83 mm, as specified by the Fluent manual.

A standard pressure-based solver was used to solve steady state solutions, using a SIMPLE pressure-velocity coupling. The flow was first converged using first order upwind discretization for all the variables. Upon convergence, they were gradually switched one at a time to second order upwind methods, and allowed to converge in between.
The $k$-epsilon turbulence model is only accurate at describing fully turbulent flows. Near the wall, a boundary layer forms where the Reynolds number is much lower than the bulk flow, and the fluid cannot be considered fully turbulent in this region. To deal with this, wall functions are used to model the flow in these regions. A simple description of turbulent boundary layers is dictated by the law of the wall, which shows how dimensionless velocity changes with respect to the distance from the wall, in dimensionless wall units. The standard wall functions chosen are valid up to the log-law region, with $y+$ values in the 30 to 100 range.

Figure 23: Refined mesh
Grid adaptation was then performed in Fluent to ensure proper capture of the boundary layer. Areas with $y+$ larger than 100 were refined, and areas with $y+$ smaller than 30 were coarsened. After the adapted case was re-converged, it was noticed that Fluent had failed to properly coarsen wall regions with $y+$ lower than 30. The figure above shows the contours of $y+$, showing low regions of $y+$ corresponding to the regions of slow flow/recirculation and/or finer tetrahedral mesh, indicating that the recirculation patterns in these regions are likely flawed due to improper shear stress calculation by the wall functions. A search through the Fluent handbook noted that the adaptation algorithm is not capable of coarsening a general 3D mesh.

![Figure 24: Contours of $y+$ on refined mesh](image)

Given more time, two options could be explored to fix this. A new mesh could be generated with the mesher which contained coarser elements in those regions. This mesh could then be refined in the regions of higher flow to give proper $y+$ at the first element, and possibly refined on the interior to better capture the large gradients in flow created by coarser elements. Another option would be to switch from standard to enhanced wall functions, which resolve the boundary layer all the way to the viscous sublayer. Starting with either the existing mesh or a newly generated one, refinement would be applied to limit the wall $y+$ between 1 and 5. While this method would provide higher accuracy than the standard wall functions, it would require many more elements and incur much additional computational cost.

**Cold Flow Results**

The final CFD results were converged numerically (residuals in the order of $10^{-5}$), as well as crudely grid converged using a different mesh type. The internal flow profiles strongly resemble those of the canonical jet in crossflow problem, validating our previous assumptions. This provides support for the assumption of flow stability mentioned earlier. As further proof, a transient simulation was run starting with imposing a small disturbance in methane mass flow on the steady state case. The flow was observed to quickly return to a steady state with no
oscillations. While these results do not guarantee the absence of instabilities, they support the notion that we can expect stable cold flow and combustion.

The figure below shows contours of methane concentration by mass in a center slice of the igniter, with the blue indicating 0% methane and the red 100% methane. Since the only other species is oxygen, these contours can also indicate the local OF ratio in different areas of the igniter. We see good mixing in the region downstream of the stream impingement, with concentration gradients evening out as the flow approaches the exit. By the time the flow crosses the exit plane, the methane concentration ranges from 32% to 35%, corresponding to the expected torch OF ratio of 2, within a 5% range.

![Figure 25: Contours of Methane mass fraction](image)

The figures below show path lines coming from the methane and oxygen inlets, colored by the local gas velocity. Looking at the methane pathlines, you can clearly see a recirculation zone attached to the back of the methane inlet, and extending about a torch cavity diameter downstream. While there is some oxygen penetration, this recirculation bubble remains very fuel rich, with an OF of about 1.5. This recirculation zone (and any others) act as flame holders once combustion is established, allowing the incoming propellants to ignite without needing to continually fire the spark plug.
Figure 26: Pathlines from the methane inlet, colored by velocity

Comparing the oxygen and methane pathlines explains the slight vertical stratification in propellant concentrations in the downstream mixing zone, as seen in the contour plot. This raises concerns for the top, ox-rich section. This section sees flow with an OF of 4-5, significantly higher than the design OF ratio for the torch. These higher OF ratios correspond to peak combustion temperatures for methane/oxygen combustion, and could cause localized melting and erosion of the wall. Due to the short duration (~500 ms) of the igniter firing, it was decided that damage was not likely and an acceptable testing risk. For designs requiring longer burns, ablative insulation around the inside of the cavity would be recommended. The lower section of the torch seeing fuel rich flow is of little concern in terms of heating, as combustion temperatures are significantly lower.

Figure 27: Pathlines from the oxygen inlet, colored by velocity
The pathlines also show that most of the methane is drawn forwards, and very little flows backwards and enters the recirculation zone around the oxygen jet and the spark plug, leaving the area around the igniter ox rich. This is favorable for ignition as the ionization of oxygen rich gas creates free oxygen and hydroxyl radicals as well as other active species, all of which are important in starting the reaction chain for methane combustion. One point of concern is that the pathlines appear to be flowing backwards through the spark gap. A closer view of the velocity field in the center plane at the vicinity of the spark plug shows this reversed flow, achieving moderate velocities in the gap before rejoining the main oxygen flow. This brings the concern that the spark may be extinguished and/or drawn away from the mixing zone, possibly leading to ignition problems. We decided it was not a large enough concern to require mitigation and design changes, it would be subject to further consideration and more in-depth analysis should testing reveal problems with igniting the torch.

Figure 28: Backwards flow through spark plug gap
Igniter Design & Manufacturing

Design Iterations

Basics of the igniter design require that it includes a port for the methane gas, a port for the oxygen gas, and an attachment point for the spark plug. For testing purposes it is also desirable to include pressure measurement port, and a suitable area to adhere one or more temperature probes to the outer surface. The igniter body itself needs to resemble a small combustion chamber with only a converging nozzle and a thick walled body to absorb the heat generated during combustion. Using an ablative inside the igniter or implementing some type of cooling would not be necessary for short duration igniter firings (~500 ms) and reduces the simplicity intended behind igniter design.

CFD analysis discussed above was used to determine optimal flow configurations and the correct placement of the spark plug so that it would occupy an oxygen rich zone. From this analysis, we saw that a crossflow configuration with the oxygen as the axial flow and methane as the crossflow was ideal. The spark plug was placed upstream of the methane inlet.

For the initial design, the igniter body was designed as a thick walled stainless steel hollow cylinder with constant inner diameter. A threaded MS port was added to one axial end and another was placed through the sidewall of the igniter. The thread for the spark plug was placed through the sidewall 180 degrees opposite the methane inlet. Both sidewall threads included a flat face to ensure that both ports would have good sealing surfaces. The spark plug would be sealed by a copper crush gasket while the MS ports would be sealed with a corner o-ring.

This initial design is all one body and does not include any type of nozzle. While this reduces complexity, we discovered that it would not allow the propellants enough residence time to fully mix and combust. Additionally, lack of a convergent nozzle would require too narrow of a combustion cavity in order to pass the mass flow desired at a reasonable torch cavity pressure.

The other consideration that we explored at the point in the design was having the methane flow either centered or offset in order to create a natural swirl to the internal flow. These two configurations for the initial design can be seen below in figure 29. The swirl configuration would be slightly more difficult to manufacture but potentially more stable. The concern with the centered flow was that the oxygen flow may perceive the methane flow as a cylindrical barrier and fluctuate between flowing around either side of the methane flow. Based on feedback from Brian Evans, we learned that jets in crossflow are quite laterally stable by nature, and that offsetting the methane flow would just result in poor mixing.
The second iteration of the design kept the main body of the igniter much the same. The oxygen, methane, and spark ports were all kept in similar positions. The primary addition to the design at this point was a converging nozzle which created a need for a second part that would have to be attached. The nozzle was a needed addition in order to choke the flow from the igniter and provide adequate back pressure within the igniter chamber.

This second part required that the outlet end of the igniter include a seal and an attachment point. To achieve this, an o-ring groove was included on the nozzle part and a pattern of 4 bolt holes was included on both parts. The nozzle profile in this first design was a series of a couple of radial curves that reached a specific nozzle throat diameter. This iteration can be seen below in figure 30.

After the second revision on the igniter we sought advice from the standpoint of both igniter function as well as manufacturing. The major feedback we got on the previous design focused on material selection, flow rates, and nozzle profile.

Our initial material selection of stainless steel was based on trying to create a large heat sink for the high temperature gases inside. Upon review, it was pointed out that stainless steel would absorb the heat but also readily melt because of low thermal conductivity. The suggestion was to use a material like copper or brass with high thermal conductivity such that the heat would be
readily transferred away from the internal chamber walls. We chose brass for the igniter material for purposes of machinability.

The other comment received on the igniter design was to reduce the flow rates for the two lines and for the overall igniter. The reason was that while our original flow rate seemed small, once combusted the heat output was excessive. Brian Evans, who was giving us feedback, has previous experience with similarly sized spark torch igniters, and noted that the igniter flame would impinge on the combustion chamber and damage it with our original mass flows. To reduce the mass flow we designed orifices in the feedlines to the igniter, and did not have to change the spark torch body itself.

On the manufacturing side, the suggestion was to revisit the nozzle profile as it would be challenging to machine accurately and it would also be readily eroded during combustion. Knowing that the brass would melt and erode during the combustion and that the converging nozzle profile does not affect the flow as strongly, the design was simplified to a converging cone and a constant area throat section. An elongated throat allows the throat inlet to erode slightly around the end of the converging section while still maintaining a constant throat area, and therefore a constant back pressure in the torch cavity.

The other addition to this final design was an extra threaded port for a pressure transducer. This port was placed downstream of the ignition point and upstream of the converging nozzle. An additional flat on the outer diameter was added for proper sealing on that port as well.

![Figure 31: Final igniter design with simple endcap and pressure transducer](image).
With the igniter being manufactured out of brass, it was a simple enough part for us to machine ourselves at the Product Realization Lab (PRL).

Starting with a solid brass rod on the lathe, the first step was to face down the outer diameter of the igniter body and nozzle section. Next, the end face was faced off and then drilled out to the desired inner diameter for the length of the chamber section. A slightly smaller diameter for the thread was maintained at the far end of the igniter body. The igniter body was then parted and faced off to the correct length.

The nozzle section was already machined to the correct outer diameter. The next step was to face off the face for the converging section. A 45 degree angle was added for the converging section of the nozzle and an o-ring groove was cut to house the selected o-ring. The Nozzle was then also parted and faced off to the right length. Progress of both parts at this point are shown in figure 32.

![Figure 32: Igniter manufacturing post-lathe](image)

The next steps to finish the parts would have been using the mill. The end of the igniter body for the oxygen line would be threaded, and then the opposite end would be drilled and threaded for the four screw holes. The same pattern would be drilled as through holes on the nozzle part. An end mill would be used to face off the sealing surfaces on the outer diameter of the igniter body. Holes would then be drilled and tapped for all of those attachment points.

In total, the manufacturing of the parts should have been completed in approximately four shop sessions (16 hours). Part of the idea behind pursuing this machining ourselves was to develop the skills in house and so that we would be able to replace parts easily if needed for future tests. The milling part of the manufacturing was not pursued when it became apparent that the rest of the test stand would not be complete before the end of the quarter.

**Test Plan**
The plan for initial testing of this final igniter design included a test setup designed to be as simple while still providing data to compare to the analysis we had done on the igniter and hopefully verify our model.

For the test we would have two separate gas cylinders with oxygen and methane. We considered including a nitrogen cylinder to provide a purge gas post test, but decided that a purge would not be necessary. The line between each cylinder and the test article would include two solenoid valves for redundancy and one relief valve to relieve any over pressure. Flashback arrestors on each line would be included for safety.

The lines from the tanks to the test article would be connected together with a combination of NPT and Swagelok fittings for standard ¼” tubing. The fitting on the test article would adapt from Swagelok to an MS port male thread. The temperature probe would be placed on the outside of the test article using high temperature adhesive.

All valves and the spark plug would all be actuated remotely from a distance to ensure safety. This is the aspect of the test setup that was not doable within the final weeks of the quarter. This was largely the case because none of the team members had the right prior experience to implement such an electrical and data acquisition system quickly. Driving a spark plug was another area where some development would be required.
Thrust Chamber Assembly

The design of a regeneratively cooled chamber assembly is largely driven by two things- the available manufacturing capabilities, which determines possible material combinations, chamber shape, and channel geometries, and a heat transfer analysis, which determines what materials and geometries will provide the desired cooling.

Chamber and Channel Design

Regen Channel Design Background

The goals of any regen channel design is to maximize conduction to the cooling fluid, cool the entire chamber evenly, keep wall temperatures below dangerous temperatures, maintain structural integrity, and minimize pressure loss of the fluid.

Because the throat experiences the highest heat flux, most regen chambers have the channels start part way down the nozzle and then flow past the throat to the rest of the chamber. Sometimes the channel starts at the very bottom of the nozzle, but this is typically not necessary since the heat flux downstream in the nozzle is lower and can be handled with a slight ablative or large heat sink.

![Cross-Sectional View of a Thrust Chamber along Axial Direction with Regenerative Cooling](image)

The basic theory behind the regen is that the hot gas heats the inner wall, which transfers this heat to the regen liquid. The outer wall typically provides the structural support necessary to contain the chamber pressure. Because each wall serves a different purpose, often two different materials are used. The inner wall is often made of a material like copper that has a high conductivity and can withstand quite high temperatures, while the outer wall would be made of a strong material like steel or inconel.

Traditionally the main designs for the cooling channels for the chamber assembly have either
been an outer and inner liner separated by a corrugated sheet, an outer liner with circular tubes brazed on the inside, an inner liner with channels milled out with an outer liner around, or an inner liner with helical channels around it.

Which of these is chosen often depends on the manufacturing technologies available. Forming circular tubes and then braising them to an outer liner accurately can be a significant challenge and expense, which is why this design is not often chosen by university teams. Milling channels into an inner piece of metal and braising it to an outer sheet is quite a bit easier. However, this still involves certain difficulties, and often the easiest thing to do is make the chamber a single piece of metal and cut the cooling holes directly into the wall.

Combustion Chamber Shape Background

The design also depends heavily on the overall shape of the combustion chamber and how many pieces are used. The three combustion chamber shapes typically used are shown below: spherical, nearly spherical, and cylindrical.

As Huzel and Huang explains, in theory, the only thing that matters for a chamber is the volume. However, long chambers can have higher pressure losses, masses, and impose other size
constraints when assembled with the rocket stage. Short chambers don’t have enough mixing volume and therefore often have low combustion efficiencies. Spherical designs have the best surface area to volume ratio, which keeps mass low and decreases the cooling surface area. However, they are hard to manufacture and more likely to exhibit instabilities, so chambers closer to cylindrical chambers are typically used.

The combustion chamber length is typically chosen to ensure full combustion and mixing, and for various propellant combinations and chamber geometries, standard characteristic lengths, $L^*$, can provide a starting point.

### Table 4-1 Typical combustion chamber characteristic length ($L^*$) for various propellant combinations.

<table>
<thead>
<tr>
<th>Propellant Combination</th>
<th>Combustion Chamber Characteristic Length ($L^*$), in.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chlorine trifluoride/hydrazine-base fuel</td>
<td>20-35</td>
</tr>
<tr>
<td>Liquid fluorine/hydrazine</td>
<td>24-28</td>
</tr>
<tr>
<td>Liquid fluorine/liquid hydrogen (GH$_2$ injection)</td>
<td>22-26</td>
</tr>
<tr>
<td>Liquid fluorine/liquid hydrogen (LH$_2$ injection)</td>
<td>25-30</td>
</tr>
<tr>
<td>Hydrogen peroxide/RP-1 (including catalyst bed)</td>
<td>60-70</td>
</tr>
<tr>
<td>Nitric acid/hydrazine-base fuel</td>
<td>30-35</td>
</tr>
<tr>
<td>Nitrogen tetroxide/hydrazine-base fuel</td>
<td>30-35</td>
</tr>
<tr>
<td>Liquid oxygen/ammonia</td>
<td>30-40</td>
</tr>
<tr>
<td>Liquid oxygen/liquid hydrogen (GH$_2$ injection)</td>
<td>22-28</td>
</tr>
<tr>
<td>Liquid oxygen/liquid hydrogen (LH$_2$ injection)</td>
<td>30-40</td>
</tr>
<tr>
<td>Liquid oxygen/RP-1</td>
<td>40-50</td>
</tr>
</tbody>
</table>

With a cylindrical chamber, the easiest way to make the cooling channels is through methods like EDM - blasting a hole straight through the wall. This is incredibly accurate and is not length constrained like a mill often is. The nozzle can either be held in the same cylindrical chamber and held in place with a retaining ring, or bolted to the cylindrical chamber separately. For the former, the nozzle would typically cooled ablatively. The latter is more mass efficient, but adding regen channels to the nozzle can be quite a challenge given that most machining processes can only make straight holes.

### Additive Manufacturing of Thrust Chamber Assembly

As described above, one the most difficult part of a combustion chamber is often its manufacturing. Each part of the combustion chamber made separate from the others, must be attached to the others, which adds weight and failure points. And as mentioned, most manufacturing processes can only make straight holes, which can make regen difficult.

Additive manufacturing is attractive because it can allow the creation of almost any geometry and of the entire combustion chamber assembly as a single part. Moreover, AM allows intricate geometries at a very small scale, which makes it very attractive for small engines. Usually, Direct Laser Metal Sintering is the process used, but it is also possible to etch pieces of sheet metal and then diffusively bond them, or to use a different 3D printing process. Each has its pros and cons.

However, AM does have some as of yet unsolved issues. The insides of the cooling channels
usually cannot be post machined and so are left with the surface finish they have after manufacturing. This surface is typically quite rough for flow purposes and leads to unacceptable pressure loses. Additionally, not all desired materials can be 3D printed, such as copper. Other AM processes can utilize copper and other materials can suffice, but this must be taken into account beforehand. Additionally, the AM processes often result in materials with different material properties. For example, most 3D printed metals are weaker than their equivalent. None of these challenges are impossible to overcome, and many are working on devising new strategies to deal with these issues.

Our Chosen Chamber Design

The design of the chamber was driven by a variety of factors including basic first order analysis, manufacturing technique, typical geometry for coolant passages, and geometry required by connection points for the injector, igniter, and fuel lines. The primary driver in the design is the manufacturing capability provided by AM as flow passages could be of variable size and be routed nearly anywhere through all parts.

The material that would be selected for AM of the chamber and injector assembly is Cobalt Chrome. This is a particular alloy available from multiple DMLS providers that possesses relatively high thermal conductivity and good strength properties. Thermal conductivity for the chamber is critical to the chamber in particular so that heat is readily transferred from the inside of the chamber to the coolant. Typically copper would be employed for the internal section of the cooling channels and a stronger material like steel would be used for the outer constraint. In this case as it all needs to be one material, Cobalt Chrome will be used for the entire piece.

Knowing the material properties of Cobalt Chrome, the approximate thickness of the inner wall was calculated using standard pressure vessels equations. With relatively low chamber pressure (250 psi), good material yield strength (128 ksi), and a reasonable chamber radius (2 inches), the thickness needed for the inner wall was only 15 thousandths of an inch (thou) with a safety factor of 2. Printing capabilities actually limit thickness to greater than 20 thou and so this number had to be bumped up. The wall thickness chosen was 30 thou to provide enough margin but not be overly thick for adequate heat transfer through the wall.

One concern brought up in using a highly conductive material with a relatively low heat capacity coolant was nucleate boiling inside the channels. This is one area where our heat transfer analysis should be further refined to assess the state of the coolant. If nucleate boiling is a problem, choosing a material with a lower heat conductivity might be a more suitable choice.

The overall geometry for the chamber determined from general sizing of the overall rocket and of the other components that would need to interface with the chamber. The inner diameter of the chamber was chosen as 4 inches and the length was chosen as 6 inches in order to allow for complete combustion. The nozzle expansion ratio was calculated for perfect expansion from nominal chamber pressure at 250 psi to ambient pressure at sea level. The nozzle profile was created as a parabolic approximation of a bell profile, with 60% of the length of an equal cone.
The geometry of the cooling channels themselves depended on factors like aspect ratio, the number of channels, and the minimum diameter that they would need to span. Aspect ratio for cooling channels in terms of length to width is typically high and we chose an aspect ratio of 3 to start with. This is a number that should also be further evaluated by examining the heat transfer characteristics. Then, knowing that the throat diameter and the channel width desired, we determined that 24 cooling channels would be the maximum that could be fit without compromising the strength of the web structure. The web as well as the outer ring of the chamber are both parts that should be evaluated for structural integrity via FEA.

With the 24 channels, the throat diameter was fully encompassed but with a constant channel profile the cooling for rest of the chamber was somewhat sparse. In creating cooling channels there is a tradeoff between pressure losses and coolant coverage. Changing the profile of the cooling channels allows for better coverage of the entire chamber but also induces greater pressure losses due to expanding and converging areas along the flow path. In most traditional manufacturing methods, variable channel profiles are highly challenging or impossible to manufacture. In our case, with AM, creating variable profiles is simple and easy to do. The recommendation from industry on this was to take advantage of variable geometry because of our manufacturing process. It is still questionable whether or not the increased cooling along the chamber walls is indeed needed and worth the pressure loss. Heating at the nozzle throat is the most intense and so cooling at that point is much more critical than at other points along the chamber or the rest of the nozzle. Revisions showing the constant and variable profiles are shown in figure 33.

With the overall geometry and the channel geometry determined, the rest of the design centered on attachment points for the injector, igniter, and coolant inlet. For the coolant inlet, the cooling channels were extended halfway down the nozzle profile and separate inlets for each channel were cut through the nozzle to an outside manifold. The outer manifold was created to be a cylindrical profile wrapped around the full circumference of the nozzle with a single attachment.
point for the coolant line. This manifold design is similar to designs we saw in our research and from our experience in industry.

Attachment for the injector at the top of the thrust chamber required both compression and a sealing surface, which were incorporated via a bolt pattern and a single o-ring groove outside of the cooling channels. We are unsure of how much sealing should be done around cooling channels such as these and believe that some sort of seal might be needed on the inside of the channels as well. The problem raised by placing an o-ring on the inside is having enough thickness for placement and the resulting distance that the channels must be from the inner wall. In our first estimation, our conclusion was that sealing on the outside would be most critical and that sealing on the inside would have to be neglected or achieved in another way. On the opposite side of this attachment is the same bolt pattern on the injector assembly and matching cooling channels that transition into the injector manifold.

The igniter attachment simply required a hole where the ignition flame could enter the chamber at the appropriate height and angle to light the incoming fuel and oxidizer. The exact location for the igniter was not determined, but the general geometry required was incorporated. A small hole was placed through the outside of the chamber wall for this purpose. If the igniter was to be manufactured separately, a seal and an attachment of some sort would also be needed. this could be achieved with an o-ring and a couple of bolts. The alternate possibility is to simply print the igniter as one piece with the chamber. In either case the hole required does present a slight complication to the cooling channels, particularly in a variable profile case. The channels must be either discontinued or somehow routed around the hole. Again, the exact geometry was not created for this because it would be affected by a number of factors we could not exactly determine in a first order design.

The design presented here is for the regeneratively cooled case. Our first chamber designs would not include cooling channels and would instead be thick walled ablative chambers that would help us first confirm the heating profile of the chamber. While much of the geometry described above would be absent in this design, the general sizing and attachment points would still be valid. Much of the reason behind doing the detailed design of the regen chamber up front was to evaluate manufacturing constraints, better define cooling for the heat transfer code, and to effectively plan ahead for the design of current and future iterations of the injector, igniter, and chamber.

Heat Transfer Analysis

Typically, the goal of the heat transfer analysis is to determine what geometry and material properties are necessary to prevent the liquid from boiling or cavitating before the injector. Gases have far lower heat capacities than liquids, so they are much worse at removing heat from the chamber wall. In most cases, if cavitation occurs, a hot spot will appear in the chamber wall, resulting in a burn through, and engine failure.
1-D Analysis

While it is possible to develop a full CFD model to analyze the heat transfer in the chamber, as proved by the igniter modeling this can be quite computationally expensive and difficult to set up, and is therefore not a good first step when analyzing multiple designs. A 1-D heat transfer analysis is much simpler, but provides results that are accurate enough for top level design decisions.

\[
\dot{Q} = h_g (T_r - T_{wg}) A_g = \frac{k_{cn}}{t_w} (T_{wg} - T_{wc}) A_g = \frac{h_c}{t_w} (T_{wc} - T_c) A_{liq} + q''_{fin} A_{fin}
\]

The 1-D heat transfer analysis begins with a general heat transfer equation like this one for square channels milled into an inner liner. It says that the convection from the hot combustion gasses to the chamber wall equals the conduction through the wall equals the convection from the wall into the regen liquid. The nomenclature is as follows:

- \( Q_{dot} \): Heat Flux
- \( T_{aw} = T_r \): adiabatic wall temperature
- \( T_c \): temperature of the coolant
- \( T_{wg} \): temperature of the wall touching the combustion gas
- \( T_{wc} \): temperature of the wall touching the coolant
- \( \eta_{fin} \): fin efficiency
- \( k \): conductivity of metal of regen wall
- \( N \): number of cooling channels
- \( A_g \): surface area of the chamber that is being heated
- \( A_{liq} \): surface area of the cooling channel
- \( t \): thickness of the wall
- \( h_g \): convection coefficient of the combustion gas with the chamber wall
- \( h_c \): convection coefficient for the coolant liquid with the chamber wall

The combustion gas properties are calculated using CEA using the general engine parameters as inputs. However, while CEA gives values for the combustion chamber and the nozzle, we were unable to calculate how the properties changed as you went along the chamber or along the nozzle. It may have been accurate enough to interpolate or assume the values remained constant, but we did not have enough time to determine if this was the case.

The methane liquid properties were estimated based on online sources such as Chemistry webbook and REFPROP. Our values were never validated, so we will not list them here.

With these properties, we could calculate the convection coefficient for the combustion gas using the Bartz Correlation for heat conduction across turbulent boundary layers:
Then for the coolant coefficient, a Seider-Tate is standard:

\[
h_g = \frac{0.026}{d_i^{0.2}} \left( \frac{C_{p,g}^{0.2}}{P_{r_g}^{0.6}} \right)_0 \left( \frac{P_c}{C^*} \right)^{0.8} \left( \frac{A_t}{A} \right)^{0.9} \sigma
\]

\[
\sigma = \left[ 0.5 \frac{T_m}{T_c} \left( 1 + \frac{\gamma - 1}{2} M^2 \right) + 0.5 \right]^{-0.68} \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{-0.12}
\]

For each of these, it is important to verify the coefficient depending on the units you choose.

Using these equations we got convection coefficients that were on the order of 1000 W/m^2, which is where they should be. However, we did not progress far enough with the analysis to be able to validate if our analysis was correct.

**Regen for LOX/Methane Specifically**

While we did not test this, we were told quite often that Methane is not always the best coolant for a regen engine. The reasoning was that standard methane does not have tremendous heat capacity and therefore it can be quite difficult to get liquid methane all the way to the injector. However, this does not mean Methane cannot be used as a coolant. One example is that supercritical methane can be pumped to a high enough pressure that it can remain liquid-like for the entire length. Additionally, a SpaceX engineer pointed out that if the location where methane starts nucleate boiling can be controlled, the convection coefficient increases dramatically and therefore the boiling methane is a fantastic coolant. The issue with the boiling methane is that it is difficult to accurately model and determine when it will transition to all gas, which will drastically affect the convection properties.

Another possibility for a LOX/Methane engine is to use LOX for the regen fluid instead of methane. While most people do not like heating oxidizers, there was tremendous research in the 60s and 70s that showed that this can be done safely. They found LOX was as good a coolant as methane and other fuels, in fact better under certain conditions.
Conclusions and Next Steps

After pursuing the development of a LOX/LCH4 regeneratively cooled engine over the course of this quarter, our conclusion at this point in time is that development should be put on hold until a later date. The decision behind this is twofold. For one, the current team has developed other responsibilities and priorities for spring quarter and would not have the time commitment to make significant progress. We have also realized the timeframe of this project has likely increased beyond our current team’s enrollment at Stanford and see a need to recruit a larger and possibly younger team at the start of the next academic year. Despite this, we have learned a significant amount about the challenges in developing a liquid rocket engine and the injector and igniter assemblies in particular. Our goal in this document is to provide the full extent of the work we have done, the challenges we encountered, and our suggestions for what next steps are necessary. We hope that what we have done this quarter will enable a team to pick up our work at later date and pursue practical next steps.

When we realized that we would not continue development at the end of this quarter, we shifted our focus to the injector and igniter and trying to get positive test results for each. For the injector, this meant getting to a successful waterflow test. Our challenge on the injector for much of the quarter was in getting a testable assembly that would not leak when printed as plastic parts. Our final effort was going to include printing the parts on a more advanced printer in order to help solve some tolerance issues and then to seal the part using acetone vapor to alleviate leaking. Unfortunately, the timetable for printing these parts on the good printer was moved past the end of the quarter and so we decided to forgo that test. We did test the process of using acetone vapor to seal the parts and encountered some issues, but believe that the process might be plausible to use if better refined.

On the igniter, we pursued manufacturing a brass body igniter that we would be able to test with a simple test stand and gas fuel and oxidizer. The fact that we could test this component with gas propellants instead of liquid propellants made testing this much more feasible on campus. The design and manufacturing of the part itself was also relatively simple and quick to do. Our goal was to have the part made and tested before the end of the quarter, but we were not able to make that goal. We started manufacturing but stopped when we realized that the test stand would not be able to be completed before the end of the quarter. Lead time on components like valves contributed to this, but we were also not comfortable making a safe, remotely-activated test setup in the short timeframe remaining. Given more time, we believe that the design we have is testable and the proposed test stand could be readily made.

Despite encountering challenges on successful tests of both the igniter and injector, we have learned a lot about liquid engines, important characteristics for specific components, and some of the unique challenges behind a LOX/LCH4 engine that’s regeneratively cooled.
The section of this report on injectors includes a good discussion on different types of injectors, some of the reasons to choose each type, and the characteristics that are important for the pintle design that we chose. Our experience and lessons learned in attempting to waterflow our injector while using rapid prototyping tools is also documented. We chose a pintle injector for both ease of manufacturing, lower susceptibility to combustion instabilities, and it’s proven use for a LOX/LCH4 engine. Our analysis gave us proper sizing for the pintle and CFD analysis helped confirm flow patterns within the pintle. In talking with industry experts, we were advised to develop the pintle from the standpoint of testing instead of further analysis and thus pursued that avenue. Through testing, we learned that creating a fast and efficient rapidly prototyped testing plan is much easier said than done and that development for the injector alone could be a time-intensive process.

For the igniter we were able to pursue a much more analytical approach to design while using CFD to predict the mixing and flow patterns within the igniter chamber. These results gave good insight into important parameters and how to go about the design, but the analysis is still likely far from complete. In this case, we’ve found that refinement of the model and the mesh is a project that could take multiple quarters before having a result that could fully predict performance. On the design side, we learned about sizing of the igniter flame, material selection, and designing for manufacturing.

Overall, we learned that taking on the development of a liquid regeneratively cooled rocket engine is a significant and time-intensive endeavor. The initial sizing we did for the whole rocket clearly demonstrated the performance increase for liquid propellants over gas propellants when looking at a full rocket system. Therefore, for application to an actual launch vehicle it is critical that the engine developed is liquid. What our sizing did not reveal is that the increase in required development effort for a liquid engine may be just as dramatic as the performance increase. Handling cryogenic propellants presents safety concerns and using liquid propellant introduces considerations like regenerative cooling, two-phase flow, cavitation, and propellant storage.

A problem we discovered specific to liquid methane is obtaining and storing methane in a liquid form. Liquid methane is challenging to work with because it must be maintained at low temperature such that it will not boil off. Creating liquid methane cannot be achieved by simply compressing methane gas, as the gas must also be cooled. In industry, creating liquid methane is done but often requires complicated equipment and is typically done in large quantities. Ordering liquid methane is possible but requires ordering amounts much larger than what is needed for a small test article. The challenge is then how to liquify methane as part of the test setup.

From our research, we found that liquefaction of methane is typically done in three ways: cooling with another cryogenic liquid, compression and expansion through a CD nozzle, or using a cryogenerator that employs a Stirling cycle. The second two options here both require expensive and complicated machinery that is used in industry but is not likely to be practical for a small scale system. Our best option therefore is to cool using another cryogenic liquid. This is something that has been at Purdue as part of their LOX/LCH4 testing.
From our contacts at Purdue, we were able to obtain reports on what they had done in developing their own capability for liquid methane. Their setup used liquid nitrogen to cool gaseous methane using a cylinder inside of a cylinder approach. The setup is fairly straightforward but would probably require a fair amount of time and effort to recreate successfully. Purdue’s team spent multiple quarters in just developing this system.

The setup (shown below) includes two tanks with one inside of the other, tubing lines and a coil to feed methane gas in and through the outer tank to be cooled, a filter to prevent frozen particles from getting in the lines, and a pressure regulator on the LN2 tank. This setup includes a couple of challenging aspects. For one, the LN2 is capable of freezing the methane and so pressure (and thus temperature) of the LN2 must be closely monitored. Also, enough methane must either be created before the test and stored or the mass flow rate during the test must be met. Purdue then investigated an alternative method using heat exchangers, which is discussed in the reports mentioned at the beginning of this document.

Figure 34: Purdue’s methane liquefaction setup
The timeframe required for the development, design, and analysis for each component is likely much longer than we initially anticipated. Even choosing to focus on just the injector and igniter with an ablative chamber was a lot to try to cover during one quarter. It would be easy to see any one component taking one or more full quarters to fully analyse and test to a working design. Developing the test stand for the engine or the liquefaction process for methane are also both projects that could take multiple quarters to get right. Obtaining the right support and advisement for developing a liquid test stand would also be a further delay.

One of the area of most resistance we encountered throughout the project was in trying to use cryogenic liquids on campus. There is currently no infrastructure in any of the ME or AA labs and professors are highly reluctant to introduce cryogens into their lab space or elsewhere on campus without prior experience and adequate safeguards. At this point, if a liquid stand was going to be made it would need to be a mobile test stand that could then be transported to a facility like FAR in Mojave, CA. This could be a solution but it also prevents the testing frequency that would be needed to really develop the engine.

One of the paths forward on this project would be to continue in much the same vein as what we’ve done so far. There is certainly more work to be done in developing both the injector and igniter. The ablative chamber and methane liquefaction process would also be parts to be pursued. The main concern here is that resistance to testing is likely to persist.

The alternate forward that we see as a good potential solution is using gaseous propellants instead. Professors have better familiarity with gas propellants and are therefore more likely to consider supporting a gas engine. Gas propellants also eliminate many of the safety concerns with cryogenic liquids as well as the need for developing a setup for liquefaction of methane. Starting with a gas engine would require an alternate injector design but would still teach a fair amount about the design of an ablative chamber and the igniter. We believe that this could be a good stepping stone towards bipropellant engines and later development of a liquid engine.

The plan for spring quarter 2014 is for the team to focus on the rocket project being pursued in AA284 and to develop a knowledge base within the larger rocket team in SSI through the ACS and P-ref projects. These projects focus on using solid rocket motors but will give younger students a good background in basic rocketry that will enable them to be more involved should the development of this engine continue next fall.
References

General:


AA284A/B Course Notes


Injectors:


Igniters:

ANSYS Meshing User’s Guide,

http://quest.arc.nasa.gov/people/journals/space/boyd/09-24-98.html

Bowman, C., ME 372 course reader


Chamber Design and Analysis:


Shoenman, L. “Low-Thrust Isp Sensitivity Study,” Aerojet, 1982
Appendix 1: Rocket Sizing code

Algorithm walk - through

We wrote a code to use for the initial design of the rocket which, given the desired mass and thrust for the rocket, calculates performance, component masses and lengths, and other general properties. It is very simple and general, which makes it a powerful tool for exploring the design space for the rocket. It is not intended to be a thorough analysis, it is a starting point. Nevertheless, it helped us understand how the different general design parameters of the rocket affect each other to shape the overall performance. Below is a rough description of the code function and key equations, followed by the commented source code.

The main functionality After reading all the rocket parameters, material/fluid properties, the code calls CEA to calculate the Isp and engine throat mass flux. Since the thrust is fixed, the code can calculate a propellant mass flow and throat diameter. Then it models the rest of the engine as a cylindrical pressure vessel with a safety factor, with the length determined by the L* parameter (see Huzel and Huang for explanation), using the following relations:

\[ T = n_{\text{thrust}} m \text{ Isp } g_0, \quad L^* = \frac{V_{\text{chamber}}}{A_{\text{throat}}}, \quad \sigma_{\text{yield}} = \frac{P_{\text{chamber}} r_{\text{chamber}}}{t_{\text{chamber}}} \times SF \]

The nozzle is then calculated as a conical profile with a set thickness with a set half-angle, beginning area given by the throat area, and exit area given by the expansion ratio. Valve mass is estimated to be a set fraction of the engine weight. An additional margin is applied to the final engine mass, to account for other sources of engine mass, such as flanges, manifolds, support structures, and regenerative cooling channels.

Next, masses and lengths of structural components such as nose cone and airframe are calculated. Airframe weight is approximated by an aluminum cylinder the diameter of the rocket and 4 times as long, to cover the engine, the spaces between tanks, and the avionics/parachute bay. Nosecone is a cone of set half angle the diameter of the rocket. Together, these make the miscellaneous masses.

Next, the code calculates the propellant tanks and pressurization system iteratively. It updates the burn time and calculates component weights until the rocket’s total mass matches the input value.

First, the total propellant mass is calculated using the mass flow, that iteration’s burn time, and a propellant utilization factor to account for propellant boiled away or left in the tank at the end of burn. The propellant is divided into oxygen and methane masses, and required tank volume is calculated by density. The volume of helium pressurant is estimated by assuming polytropic expansion from the high pressure helium tank, with an end condition of the propellant tanks.
filled with pressurized helium and nothing else. The pressurant tank is assumed at methane tank pressure, since it is larger than oxygen tank pressure.

\[ V_{\text{HeTank}} \rho_{\text{initial}} = V_{\text{O2Tank}} \rho_{\text{final}} + V_{\text{HeTank}} \rho_{\text{final}} + V_{\text{CH4Tank}} \rho_{\text{final}} \]

\[ \frac{\rho_1}{\rho_2} = \left( \frac{P_1}{P_2} \right)^{1/n}, \varepsilon_{\text{O2}} = \left( \frac{P_{\text{O2Tank}}}{P_{\text{HeTank}}} \right)^{1/n}, \varepsilon_{\text{CH4}} = \left( \frac{P_{\text{CH4Tank}}}{P_{\text{HeTank}}} \right)^{1/n} \]

\[ V_{\text{HeTank}} = V_{\text{O2Tank}} \varepsilon_{\text{O2}} + V_{\text{HeTank}} \varepsilon_{\text{CH4}} + V_{\text{CH4Tank}} \varepsilon_{\text{CH4}} \]

Having determined the tank volumes, they are modeled as cylinders with spherical endcaps the diameter of the rocket, with the thickness determined by the same stress equation as shown above for the combustion chamber. The feedline is modeled as a tube the length of both propellant tanks.

\[ m = \rho V_{\text{feed}} A_{\text{feed}} \]

The diameter is calculated to fit a given feedline velocity, which ensures that the pressure drop due to feedline losses is not affected by changes in mass flow. Finally, the burn time estimate is updated as a linear scaling involving the difference between the total mass and the engine and miscellaneous masses (which are burn time independent). The burn time converges rapidly from an initial guess.

\[ t_{\text{burn}}^{i+1} = t_{\text{burn}}^i \left( 1 - \frac{M_{\text{total}} - M_{\text{eng}} - M_{\text{Misc}}}{M_{\text{target}} - M_{\text{eng}} - M_{\text{Misc}}} \right) \]

\[ \Delta V = \eta_{\text{thrust}} I_{\text{sp}} g_0 \ln \left( \frac{M_{\text{total}}}{M_{\text{total}} - M_{\text{prop}} \eta_{\text{util}}} \right) \]

Once the final configuration is determined, delta-V is calculated and outputs are plotted and displayed. For more information see Huzel’s and Sutton’s texts.
function [deltaV,Ltotal]=rocketsize(Pc,OF,Aratio,Thrust2weight,Drocket)
% Requires NASA CEA program to be installed in order to get combustion
% properties
%
%MEOP = Max expected operating pressure
%MAWP = Max allowable working pressure = 1.5*MEOP

% Press tanks right before launch, necessary to keep prop densities
% high and prevent 2 phase flow during blowdown due to pressure release

%% Declare parameters
grav=9.81;

% Margins:
Tank2CCdP = .4; % dP from tanks to combustion chamber as percent of chamber pressure,
assumed 10% tank to injector face and 10% injector face to CC
MarginProptank= .25; %
MarginHetank= .25; %
Marginengine= .6;
Safetytank= 3; % Burst pressure/MAWP
PHratio= 6; % Ratio of He tank pressure to highest propellant tank pressure, methane
RegenDP= 120*6.8948; % Extra pressure to run regen cooling

% CEA outputs
%OF= 3; % CEA input
%Cpc=250; % psi
%Aratio= 6;
rperf=CEA_Win2('O2(L)', 'CH4(L)',OF,Pc,Aratio,90,110,7);
Pchamber=rperf.P_chamber/1000;% kPa
Pexit=rperf.P_exit/1000
Vexhaust= rperf.Isp_actual*grav; %m/s Isp*g, Pe= 7psi, corrected for Pa at 50,000 ft lowest
without risking separation at ground test
Mfluxthroat= rperf.u_throat*rperf.rho_throat; %kg/s/m2 , mass flux through throat

% Materials & Fluid properties
Pmethtank=Pchamber*(1+Tank2CCdP)+RegenDP; % kPa
Ploxtank=Pchamber*(1+Tank2CCdP); % kPa
Rholox= 1154; % kg/m3 At saturation for ambient P
Templox= 90.2; % K Saturation temp for ambient P
Rhometh= 422; % kg/m³ Supercritical 311.3
Tempmeth= 112; % K Saturation temp for tank P
Temproom = 273; % K
Sigmacryo= 352; % MPa strength at cryo temp, SS 316
Sigmaroom= 352; %
Sigmahot= 165; % MPa strength of combustion chamber, SS 316 @ 500 C
Rhoinsul= 9.61;
Rhotank= 7990; % kg/m³, density for all pressure vessels, SS steel
Rhotankal= 2800; % kg/m³, density for all pressure vessels, SS steel
Rhostruct= 2700; % kg/m³, density for nose & structure, Aluminum
Phelium= PHeratio*Pmethtank;
RhoHe= Phelium*1000/(Temproom*8.314/.004); % kg/m³
polyHe= 1.4; % Polytropic expansion constant for helium

% Efficiencies
Thrustefficiency= .93; % Theoretical to Actual thrust
Ullagefrac=.1; % Fraction of full tank for ullage
Proputilization= .85; % Fraction of stored propellant used for combustion
Valvefrac=.1; % fraction of total engine mass composed of valves
Installfactor= .05; % Fraction of total rocket mass composed of not-accounted for parts (welds, screws, etc)

% Rocket Parameters
Drocket=Drocket*.0254; %8*.0254; %m
%Thrust2weight= 5;
tburnguess= 8; %s
Thrusttarget= 1000*4.4482; %N
Noseangle= 20; %degrees
Thickstruct=3/16*.0254; %m, thickness of nose and structural elements

% Engine Parameters
Lstar= 30*.0254; %m characteristic length of combustion chamber
Achamber2Athroat= 9; % guess
Nozzangle= 15; %degrees
Thicknozz= .5*.0254; % m guess
Rhonozz= 2230; %kg/m³ graphite
Thickablate= .5*.0254; %m guess
Injthick= .5*.0254; %m

% Misc Parameters
Insulthick= 3/8*.0254; %m thickness of insulation on cryotanks
Mparachute= 5.5; %kg, Logan's estimate for 200lb rocket falling at 5m/s
Mpayload= 1/2.2; %kg, Camera and shit
Mavionics= 5/2.2; %kg, guess for circuits, sensors, battery
Vfeed = 5; % m/s, max fluid velocity in feedline for sizing

Mtarget = Thrusttarget/grav/Thrust2weight;
%Mtgt = 2.205*Mtarget

Mnose = (Drocket^2*pi/4)/sind(Noseangle)*Thickstruct*Rhostruct; % kg
Mstruct = (pi*Drocket^2*4)*Thickstruct*Rhostruct; % kg connecting structure, cylinder L/D=5
Mmisc = Mnose + Mstruct + Mparachute + Mpayload + Mavionics;
Lnoose = Drocket/tand(Noseangle);

Mdotp = Thrusttarget/Vexhaust/Thrustefficiency; % kg/s
Athroat = Mdotp/Mfluxthroat; % m^2
Aexit = Aratio*Athroat;
Mnozzle = (Aexit - Athroat)/sind(Nozzangle)*Thicknozz*Rhonozz;
Achamber = Achamber2Athroat*Athroat;
Dchamber = 2*sqrt(Achamber/pi)
Lnozzle = (sqrt(Aexit/pi) - sqrt(Athroat/pi))/tand(Nozzangle);
Rhoprop = (1+OF)/(1/Rhometh+OF/Rholox); % kg/m3 average propellant density
Thickchamber = Pchamber*.001*(Dchamber/2)/Sigmahot*1.5*Safetytank; % m,
Lchamber = Lstar*Athroat/Achamber;
Mchamber = (Dchamber*Lchamber+Achamber-Athroat)*(Thickchamber*Rhotank+Thickablate*Rhonozz); % kg
Minj = Achamber*Injthick*Rhotank;
Mengine = (Mchamber + Minj + Mnozzle)/(1 - Valvefrac)*(1 + Marginengine); % kg total engine mass
Mvalve = Valvefrac*Mengine;

Msubtot = (Mengine + Mmisc)/(1 - Installfactor);

if (Msubtot > .8*Mtarget)
    error(['Rocket too big, Mengine = ' num2str(Mengine)]);
end

%% Loop
Iter = 60;
tol = .001;

tburn = zeros(1,Iter);
%Mtotal = zeros(1,Iter);
tburn(1) = tburnguess;

for i = 1:Iter-1
    Mprop = Mdotp*tburn(i)/Proputilization;
    Mmeth = Mprop/(1+OF);
    Mlox = Mprop - Mmeth;
    Mvalve = Valvefrac*Mengine;
    Mchamber = (Dchamber*Lchamber + Achamber - Athroat)*(Thickchamber*Rhotank + Thickablate*Rhonozz);
    Minj = Achamber*Injthick*Rhotank;
    Mengine = (Mchamber + Minj + Mnozzle)/(1 - Valvefrac)*(1 + Marginengine);
    Msubtot = (Mengine + Mmisc)/(1 - Installfactor);
end
\[ \text{Vmeth} = \text{Mmeth} / \text{Rhometh} / (1 - \text{Ullagefrac}); \] %m^3
\[ \text{Vlox} = \text{Mlox} / \text{Rholox} / (1 - \text{Ullagefrac}); \]

\[ \text{Lmeth} = (\text{Vmeth} - \pi \cdot \text{ Drocket}^3 / 6) / (\pi \cdot \text{ Drocket}^2 / 4); \] %m, cylinder w/hemisphere end tank, length of cylinder part
\[ \text{Lloxt} = (\text{Vlox} - \pi \cdot \text{ Drocket}^3 / 6) / (\pi \cdot \text{ Drocket}^2 / 4); \]

\[ \text{Thickmeth} = \text{Pmethtank} \cdot 0.001 \cdot (\text{ Drocket} / 2) / \text{Sigmacryo} \cdot 1.5 \cdot \text{Safetytank}; \] %m,
\[ \text{Thicklox} = \text{Ploxtank} \cdot 0.001 \cdot (\text{ Drocket} / 2) / \text{Sigmacryo} \cdot 1.5 \cdot \text{Safetytank}; \]

\[ \text{Mmethtank} = (\text{Lmeth} \cdot \pi \cdot \text{ Drocket} \cdot \pi + \text{ Drocket}^2 \cdot \pi) \cdot (\text{Thickmeth} \cdot \text{Rhotankal} \cdot (1 + \text{Marginproptank}) + \text{Insulthick} \cdot \text{Rhoinsul}); \] %kg
\[ \text{Mloxtank} = (\text{Lloxt} \cdot \pi \cdot \text{ Drocket} \cdot \pi + \text{ Drocket}^2 \cdot \pi) \cdot (\text{Thicklox} \cdot \text{Rhotankal} \cdot (1 + \text{Marginproptank}) + \text{Insulthick} \cdot \text{Rhoinsul}); \] %kg

\[ \text{ThickHe} = \text{Phelium} \cdot 0.001 \cdot (\text{ Drocket} / 2) / \text{Sigmaroom} \cdot 1.5 \cdot \text{Safetytank}; \] %m
\[ \text{expanmeth} = (\text{Pmethtank} / \text{Phelium})^{(1 / \text{polyHe})}; \] % Density ratio of helium expanding from Phe tank pressures to Pmeth
\[ \text{expanlox} = (\text{Ploxtank} / \text{Phelium})^{(1 / \text{polyHe})}; \] % Density ratio of helium expanding from Phe tank pressures to Plox
\[ \text{VHe} = (\text{Vmeth} \cdot \text{expanmeth} + \text{Vlox} \cdot \text{expanlox}) / (1 - \text{expanmeth}); \] %Enough to press all empty tanks to respective P tanks
\[ \text{LHetank} = (\text{VHe} \cdot \pi \cdot \text{ Drocket}^3 / 6) / (\pi \cdot \text{ Drocket}^2 / 4); \]
\[ \text{MHetank} = (\text{LHetank} \cdot \pi \cdot \text{ Drocket} \cdot \pi + \text{ Drocket}^2 \cdot \pi) \cdot \text{ThickHe} \cdot \text{Rhotankal} \cdot (1 + \text{MarginHetank}); \] %kg
\[ \text{MHe} = \text{VHe} \cdot \text{RhoHe}; \] %kg

\[ \text{Dfeed} = \sqrt{(4 / \pi \cdot \text{Mdotp} / \text{Vfeed} / \text{Rhoprop})}; \] %m^2 Diameter of averaged feedline
\[ \text{Thickfeed} = \text{Pmethtank} \cdot 0.001 \cdot (\text{Dfeed} / 2) / \text{Sigmacryo} \cdot 1.5 \cdot \text{Safetytank}; \] %m,
\[ \text{Mfeed} = \pi \cdot \text{Dfeed}^2 / (2 \cdot (\text{Thickfeed} \cdot \text{Rhotankal} + \text{Insulthick} \cdot \text{Rhoinsul})]; \]

\[ \text{Mtotal} = (\text{Mprop} + \text{MHe} + \text{Mloxtank} + \text{Mmeth} + \text{Mhetank} + \text{Mengine} + \text{Mfeed} + \text{Mmisc}) / (1 - \text{Installfactor}); \] %kg total mass
\[ \text{Ltotal} = \text{LHetank} + \text{Lmeth} + \text{Lloxtank} + 4 \cdot \text{Drocket} + \text{Lchamber} + \text{Lnozzle} + \text{Lnose}; \]

\[ \text{burntime} = \text{tburn}(i); \]
if abs(Mtotal(i) - Mtarget) < tol
  break;
else
  deltmratio = (Mtotal(i) - Mtarget) / (Mtarget - Msubtot);
  tburn(i+1) = tburn(i) * (1 - deltmratio);
end
% figure(1);
% plotyy(1:i,tbodyurn(1:i),1:i,Mtotal(1:i)*2.2);
end

%%
deltaV=Vexhaust*Thrustefficiency*log(Mtotal(end)/(Mtotal(end)-Mprop*Proputilization))-grav*burntime;
altitude=deltaV^2/2/grav;
alt=altitude*3.281/1000;
%Mend=Mtotal(end)*2.205

%% Print results
if false
    figure(1);
    plot(1:i,Mtotal*2.205);
    figure(2);
    pie([Mprop,MHe,Mloxtank,Mmethtank,MHetank,Mengine,Mmisc,Mtotal(end)*Installfactor],{'Mprop','MHe','Mloxtank','Mmethtank','MHetank','Mengine','Mmisc','MInstall'});
    title('Mass breakdown');
    figure(3);
    pie([LHetank+Drocket,Lmethtank+Drocket,Lloxtank+Drocket,Lchamber,Lnozzle,Lnose,Drocket],{'LHetank','Lmethtank','Lloxtank','LCombustion','Lnozzle','Lnose','LAVionics'});
    title('Length breakdown');

fileID = fopen('Rocket_Dimensions.txt','w+');
fprintf(fileID, '====================================
');
fprintf(fileID, 'Rocket Sizing Output
');
fprintf(fileID, '====================================
');
fprintf(fileID, 'Rocket Performance
');
fprintf(fileID, '------------------------------------
');
fprintf(fileID, 'mdot = %2.3f kg/s
', Mdotp);
fprintf(fileID, 'M_propellant = %2.2f kg
', Mprop);
fprintf(fileID, 'M_total = %3.2f kg (%3.1f lbs)
', Mtotal(end),Mtotal(end)*2.2);
fprintf(fileID, 'L_total = %2.1f ft
', Ltotal*3.281);
fprintf(fileID, 'L/D = %2.1f
', Ltotal/Drocket);
fprintf(fileID, 'Thrust = %3.1f lbs
', Thrusttarget/4.448);
fprintf(fileID, 'dV = %3.2f m/s
', deltaV);
fprintf(fileID, 'altitude = %4.1f ft
', altitude*3.281);
fprintf(fileID, '------------------------------------');
Defined Parameters

D_rocket = %2.1f in
D_chamber = %2.1f in
D_P = %0.2f %%
T/W = %2.2f in
T_Burn = %3.2f s
eff_thrust = %2.1f %%
Install Factor = %2.1f %%
O/F = %1.1f
V_exit = %4.1f m/s
AR = %1.1f
T = %3.1f K
rho_methane = %3.1f kg/m^3
P = %3.0f psi
rho_tank = %4.1f kg/m^3
sigma = %3.0f MPa
th_insulation = %1.2f in
rho_insulation = %2.2f kg/m^3
Mass Margin = %2.1f %
Ullage Fraction = %2.1f %
Propellant Used = %2.1f %
SF = %1.1f
L = %1.3f m
th = %1.4f m
M_methane = %2.2f kg
Volume_tank = %2.4f m^3
Mass_tank = %2.2f kg

CEA Performance Parameters

O/F = %1.1f
V_exit = %4.1f m/s
AR = %1.1f
T = %3.1f K
rho = %3.1f kg/m^3
P = %3.0f psi
rho_tank = %4.1f kg/m^3
sigma = %3.0f MPa
th_insulation = %1.2f in
rho_insulation = %2.2f kg/m^3
Mass Margin = %2.1f %
Ullage Fraction = %2.1f %
Propellant Used = %2.1f %
SF = %1.1f
L = %1.3f m
th = %1.4f m
M_methane = %2.2f kg
Volume_tank = %2.4f m^3
Mass_tank = %2.2f kg
fprintf(fileID, 'Mass Margin = %2.1f \%\n', MarginProptank*100);
fprintf(fileID, 'Ullage Fraction = %2.1f \%\n', Ullagefrac*100);
fprintf(fileID, 'Propellant Used = %2.1f \%\n', Proputilization*100);
fprintf(fileID, 'SF = %1.1f \n', Safetytank);
fprintf(fileID, 'L = %1.3f m \n', Lloxtank);
fprintf(fileID, 'th = %1.4f m \n', Thicklox);
fprintf(fileID, 'M_lox = %2.2f kg \n', Mlox);
fprintf(fileID, 'Volume_tank = %2.4f m3 \n', Vlox);
fprintf(fileID, 'Mass_tank = %2.2f kg \n', Mloxtank);
fprintf(fileID, '--------------------------\n');
fprintf(fileID, 'Helium Tank\n');
fprintf(fileID, '------------------------------------\n');
fprintf(fileID, 'T = %3.1f K \n', Temproom);
fprintf(fileID, 'rho = %3.1f kg/m3 \n', RhoHe);
fprintf(fileID, 'P = %3.0f psi \n', Phelium/6.8948);
fprintf(fileID, 'rho_tank = %4.1f kg/m3 \n', Rhotankal);
fprintf(fileID, 'sigma = %3.0f MPa \n', Sigmaroom);
fprintf(fileID, 'SF = %1.1f \n', Safetytank);
fprintf(fileID, 'L = %1.3f m \n', LHetank);
fprintf(fileID, 'th = %1.4f m \n', ThickHe);
fprintf(fileID, 'M_He = %2.2f kg \n', MHe);
fprintf(fileID, 'Volume_tank = %2.4f m3 \n', VHe);
fprintf(fileID, 'Mass_tank = %2.2f kg \n', MHetank);
fprintf(fileID, '------------------------------------\n');
fprintf(fileID, 'Combustion Chamber\n');
fprintf(fileID, '------------------------------------\n');
fprintf(fileID, 'D = %2.2f in \n', Dchamber/.0254);
fprintf(fileID, 'L* = %2.1f in \n', Lstar/.0254);
fprintf(fileID, 'Lchamber = %2.1f in \n', Lchamber/.0254);
fprintf(fileID, 'th_ablative = %1.2f in \n', Thickablate/.0254);
fprintf(fileID, 'rho_ablative = %4.1f kg/m3 \n', Rhonozz);
fprintf(fileID, 'rho_tank = %4.1f kg/m3 \n', Rhotank);
fprintf(fileID, 'sigma = %3.0f MPa \n', Sigmahot);
fprintf(fileID, 'P = %3.0f psi \n', Pchamber/6.8948);
fprintf(fileID, 'th = %1.4f m \n', Thickchamber);
fprintf(fileID, 'M_chamber = %2.2f kg \n', Mchamber);
fprintf(fileID, '------------------------------------\n');
fprintf(fileID, 'Engine \n');
fprintf(fileID, '------------------------------------\n');
fprintf(fileID, 'Nozzle Angle = %2.1f deg \n', Nozzangle);
fprintf(fileID, 'th_nozzle = %2.2f in \n', Thicknozz/.0254);
fprintf(fileID, 'rho_nozzle = %4.1f kg/m3 \n', Rhonozz);
fprintf(fileID, 'A_throat = %1.5f m \n', Athroat);
fprintf(fileID, 'A_exit = %1.5f m \n', Aexit);
fprintf(fileID, 'L_nozzle = %1.3f ft \n', Lnozzle*3.281);
fprintf(fileID, 'M_nozzle = %2.2f kg \n', Mnozzle);
fprintf(fileID, 'th_inj = %2.2f in \n', Injthick/.0254);
fprintf(fileID, 'rho_inj = %4.1f kg/m^3 \n', Rhotank);
fprintf(fileID, 'M_inj = %2.2f kg \n', Minj);
fprintf(fileID, 'Valve Fraction = %2.1f \% \n', Valvefrac*100);
fprintf(fileID, 'M_valve = %2.2f kg \n', Mvalve);
fprintf(fileID, 'Mass Margin = %2.1f \% \n', Marginengine*100);
fprintf(fileID, 'M_engine = %2.2f kg \n', Mengine);
fprintf(fileID, '-------------------------------------\n');
fprintf(fileID, 'Feedlines \n');
fprintf(fileID, '-------------------------------------\n');
fprintf(fileID, 'V_feedline = %1.2f m/s \n', Vfeed);
fprintf(fileID, 'D_feedline = %1.1f in \n', Dfeed/.0254);
fprintf(fileID, 'th_insulation = %1.2f in \n', Insulthick/.0254);
fprintf(fileID, 'rho_insulation = %2.2f kg/m^3 \n', Rhoinsul);
fprintf(fileID, 'th_feedline = %1.3f in \n', Thickfeed/.0254);
fprintf(fileID, 'M_feedline = %2.2f kg \n', Mfeed);
fprintf(fileID, '-------------------------------------\n');
fprintf(fileID, 'Misc \n');
fprintf(fileID, '-------------------------------------\n');
fprintf(fileID, 'L_nose = %1.3f ft \n', Lnose*3.281);
fprintf(fileID, 'th_nose = %3.2f in \n', Thickstruct/.0254);
fprintf(fileID, 'M_parachute = %2.2f kg \n', Mparachute);
fprintf(fileID, 'M_payload = %2.2f kg \n', Mpayload);
fprintf(fileID, 'M_avionics = %2.2f kg \n', Mavionics);
fprintf(fileID, 'M_nose = %2.2f kg \n', Mnose);
fprintf(fileID, 'M_struct = %2.2f kg \n', Mstruct);
fprintf(fileID, 'M_misc = %2.2f kg \n', Mmisc);
fprintf(fileID, '====================================\n');
fclose(fileID);
end