Small Rocket Engine Cooling

Last Updated: 9/12/2022

## Rocket Thermal Loads

The wall temperature of the combustion chamber is the source of one of the biggest structural vulnerabilities in the whole vehicle or stand assembly - it is one of the thinnest points of the engine while being subjected to some of the highest temperatures and resulting thermal stresses. In rocket engines, there are two primary thermal criteria which must be considered: thermal stresses and thermal weakening of the material or outright melting, in that order of severity.

Thermal stresses arise from sharp temperature gradients radially across the material and thermal expansion. In thicker sections of the nozzle, heat cannot conduct very quickly across the nozzle. As a result, the inner surface of the nozzle becomes much hotter than the outer region. This results in the inner region undergoing much more severe thermal expansion than the outer regions, resulting in a circumferential hoop stress due to the expanding inner portion being mechanically constrained by the relatively static outer portion. At particularly high temperatures with very thick nozzles, these stresses can become so severe that the yield or even tensile strength of the material is exceeded in inner sections of the nozzle, which can noticeably damage the nozzle.

The other thermal consideration which must be taken into account is the change of material properties at very high temperatures. Notably, the yield and tensile strengths of most stainless steels decrease noticeably above 500 degrees celsius, with 316 stainless steel retaining most of its standard strength up to about 600 degrees celsius[1]. Fortunately, the mechanical pressure loads on a rocket nozzle of a small size, as is applicable to this project, can be quite small. Of course, if the temperature of the metal gets hot enough (around 1600K), the material of the nozzle can melt altogether - obviously a very unfavorable phenomenon to have occurring in a rocket nozzle.

These two thermal effects can rapidly cause structural failures of a rocket nozzle altogether. This is why a key aspect of designing a liquid rocket nozzle is cooling.

## Introduction to Cooling Methods

There are many intricate methods of cooling a rocket nozzle - regenerative cooling, expander cycles, transpiration cooling, dump cooling, insulative coatings, and so on. However, for a project on a student scale, very simple, very easy to implement, and fairly cheap cooling is desirable. This narrows the wide variety of known cooling techniques down to a handful, namely: boundary layer cooling, heat sink cooling, and ablative coatings.

The most effective methods of cooling the nozzle is to coat the inside of the combustion chamber and nozzle with an ablative coating, or to add an ablative insert surface to the nozzle. Ablatives usually, but not always, consist of high-carbon materials like graphite. When these materials are subjected to the heat and high flow velocity of the working fluid, they are usually melted, sublimated, or only slightly exothermically “burned” off of the surface. When they do so, the heat which enters the ablative leaves with the ablated material rather than conducting through the ablator and into the nozzle structure. Ablatives are a fairly simple solution to thermal issues: they require no plumbing, little added structure, and are essentially ready to go after being applied to the nozzle and allowed to set. However, selecting a suitable spray- or spread-on ablative can rapidly become expensive, and machining materials such as graphite poses health hazards due to graphite dust. For these reasons, ablatives have been foregone in current iterations of the rocket design (as of September 2022).

One of the equally common, and one of the oldest, forms of cooling is boundary layer cooling, or film cooling. The concept of boundary layer cooling is simple enough: a layer of roughly ambient temperature fuel is injected from the injector plate along the surface of the combustion chamber. Oxidizer is not used because it would react with the metal wall. The boundary layer fuel (now known as “film coolant”) acts as a thermal barrier which heat from the working fluid must convect through in order to reach the nozzle wall. If the coolant is injected farther downstream from the walls of the chamber it is known as “transpiration cooling”. Boundary layer cooling is a fairly cheap and easy method of mitigating thermal dangers of the combustion chamber. However, it rapidly loses effectiveness as the coolant heats up as it travels through the nozzle. It is therefore unreasonable to assume that film cooling alone is enough to prevent thermal damage to the nozzle all the way across the axial distance of the nozzle, although it is quite effective within the combustion chamber itself. Numerical models of boundary layer cooling can be found in the boundary layer cooling modeling document[A].

The final method of cooling which is considered is heat sink cooling. Heat sink cooling does not fit neatly into what is colloquially thought of as “cooling”. No heat is removed from the structure of the engine, nor is any heat flux into the nozzle wall mitigated. Instead, a high thermal conductivity, high specific heat capacity material is added to the nozzle. In this case, the nozzle itself can be made of such a material. The principle is very simple, if somewhat brute-force in nature. If there is more material of a high heat capacity, then it takes more heat flux to heat the engine per unit temperature. Adding more mass, or just making the nozzle thicker, then means that the nozzle is able to absorb more heat before reaching some undesirable temperature. Of course, the limitation of this is that if the nozzle is made too thick, it will both become too heavy to be practical and will experience the high temperature gradients mentioned in the section above. However, for rockets which are intended to be fired for very short periods of time, such as those of this student project, this is considered a reasonable, if crude, method of mitigating thermal loads.

## Convective Heat Transfer

Heat is transferred into the nozzle walls through a process known as convective heat transfer. This is the process which usually takes place when some fluid, either flowing across the surface or flowing only through free convection, transfers heat into some solid surface. For these purposes, the temperature of the fluid is considered essentially fixed, with effects of rate of heat transfer accounted for with some constant heat transfer coefficient. This process is described by the following equation 1:

(1)

Where:

* is the area-specific heat flow rate in W/m^2
* is the total heat flow rate in W
* is the area contacting the fluid in m^2
* is the instantaneous temperature of the fluid in K
* is the instantaneous temperature of the wall in K
* is the heat transfer coefficient of the fluid in W/m^2-K

For a simple system with some total heat capacity, this law can be used to model temperature with the following system of differential equations:

(2)

(3)

Where:

* is the temperature of some object with uniform temperature
* is the total heat capacity of the object

The system can be solved analytically with some total heat capacity and total , or can be solved numerically, for example with MATLAB’s *ode45()* function. This can be used to get “ballpark” numbers, but for higher-fidelity, more “final” analyses, finite element analysis[B] should be employed to provide a plot of temperature (and exported to find thermal stresses) across the whole structure of the engine. All that needs to be obtained for these methods is the wall temperature and heat transfer coefficient.

## Heat Transfer Coefficient

In order to find the wall temperature, we can use an equation in Sutton[2] on pg. 313 to find the gas film coefficient,

(4)

Where:

* is the heat transfer coefficient of a gas in W/m^2-K
* is the coolant density in kg/m^3
* is the injection velocity in m/s
* D is the chamber diameter in m
* is the Prandtl number
* is the thermal conductivity in W/m-K
* is the dynamic viscosity in N-s/m^2

This equation requires a few values which are dependent on chamber conditions, such as the flow velocity. The flow velocity can be backed out from the mach number at some axial point in the nozzle using the velocity-to-mach relations found in the nozzle flow document[C].

Note that the Prandtl number can be retrieved from CoolProp as specified in the CoolProp document[D] using the output variable “PRANDTL”, thermal conductivity can be retrieved using output variable “L”, and dynamic viscosity can be retrieved using output variable “V”.

## References and Sources

[1] [High Temperature Characteristics of Stainless Steels](https://drive.google.com/file/d/14r85wS5sclB2xJhrSvpo3VIiAhVWI0ND/view?usp=sharing)

[2] [Rocket Propulsion Elements](https://drive.google.com/file/d/1muyScRo6bWxT6AzNZnpIaxudrqsAPFAy/view?usp=sharing)

## 

## Related Documentation

[A] [Boundary Layer Cooling Modeling](https://docs.google.com/document/d/1dI0wGCt07ZiEjnhjms6PoY4VBhTD_azOgpbDyiLKHnE/edit#heading=h.gxb1iareuwnd)

[B] [Finite Element Analysis](https://docs.google.com/document/d/1g85HHhE5HMR1voVQZ7Xsq_mlPU-9zkPZI_O_tEHfW-0/edit)

[C] [Chamber Flow Modeling](https://docs.google.com/document/d/16WyGRw5Ikb9o1UvMvrIiOay80P_ZhJnjGX2SzdEGsrk/edit#)

[D] [CoolProp Quick Reference](https://docs.google.com/document/d/1SwY_JbAcMK3dY37hVzANKK0KHyNtMjkAvfUAsUOoy1Y/edit?usp=sharing)