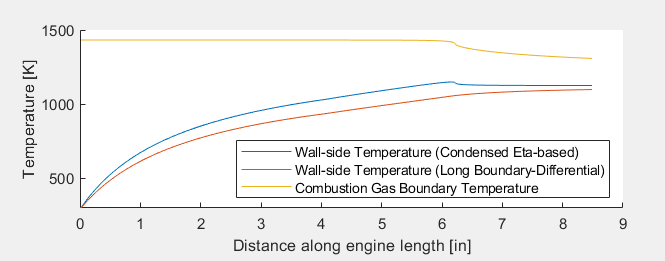
Gaseous Boundary Layer Cooling Modeling

Last Updated: 9/12/2022

## Boundary Layer Cooling Principles

Boundary layer cooling is a very simple and effective method of cooling the combustion chamber walls of a rocket. The basic idea is that the freestream working fluid is separated from the wall by the coolant-filled boundary layer. The heat from the working fluid then must convect through the boundary layer, which then in turn convects heat into the chamber wall. As the coolant travels downstream, more and more heat convects into the coolant - so the average temperature throughout the boundary layer increases with axial distance downstream. This, in turn, means that boundary layer cooling is less effective as you go further downstream. Below in figure 1 is a qualitative example of a curve of cooled wall temperature over axial distance.



**Figure 1: Qualitative Example of Cooled Wall Temperature vs Axial Distance**

When designing a liquid rocket engine, the most important thermal parameter is the heat flow or heat flux into the engine, one of the parameters of which is the temperature of the gas contacting the wall. Therefore, it is critical to determine the wall-side gas temperature, or adiabatic wall temperature, in combination with the heat transfer coefficient (see document [A], heat transfer coefficient section).

## Expanded Numerical Modeling of Wall-Gas Temperature

For the purposes of this project, the boundary layer coolant can be considered a gas upon injection, due primarily to ethane’s high vapor pressure and to a somewhat lesser extent the high freestream temperatures within the combustion chamber. This rather drastically reduces the performance of the boundary layer coolant, but at least makes modeling of the effects of cooling somewhat simpler. An outline of a numerical model can be found in the air force technical report AEDC-TR-91-1 by William Grissom[1].

For this method, certain nomenclatures must be defined. They are listed below:

* is the freestream mass flow rate per unit area, or mass flux in kg/s-m^2
* is a turbulence correction factor, unitless, where
  + is some scalar turbulence fraction, usually ~0.1
* is the total mass flow rate of the boundary layer at some point per unit circumference of the nozzle at the considered point in kg/s-m
* is the coolant mass flow rate per unit circumference in kg/s-m
* is instantaneous chamber diameter, in m
* is the adiabatic wall temperature at some axial distance in K
* is the static freestream temperature in K
* is the recovery temperature, or boundary layer side chamber temperature in K

Grissom then outlines step-by-step a method for predicting the adiabatic wall temperature across the length of the nozzle. While they are provided in the form of differential equations (e.g. ), if variables such as, for example, are treated as finite values (i.e. treat as ) and broken into many finite step sizes, the problem becomes similar to a higher-order application of Euler’s method. The equations which drive such a model are outlined below.

First, initial conditions must be set, and correction factors calculated. They are as follows:

Initial diameter:

Initial circumferential coolant flow rate:

Initial boundary layer flow rate:

Initial wall temperature:

Molar weight correction factor:

Turbulence correction factor:

Begin at some point of distance downstream. In this case, we will begin at , where is the step size, in meters, such that across each step the finite distance .

First, calculate the change in diameter at that point. This can be done using downstream distance of the nozzle inlet, or the beginning of the converging section, , that of the throat, and that of the end of the nozzle, , as well as the angle of contraction and expansion . Save the finite stepwise change in diameter as as follows:

(1)

Then, the discrete value of dx can be calculated where is the distance parallel to the contour.

(2)

Contour distance x is then calculated simply:

(3)

Current flow area is calculated using the diameter:

(4)

Freestream mass flux is calculated using the area and given constant freestream mass flow rate:

(5)

Freestream mach number is then calculated using methods outlined in the nozzle flow modeling document[B]. Next, the properties of both the coolant and the freestream gas must be retrieved. The only practical way to do so is to use software, preferably Cantera, as outlined in the Quick Reference document[C], *after* setting the respective gasses to their static conditions. Specifically, the isobaric heat capacity , dynamic viscosity , and thermal conductivity must be obtained for both the coolant gas/fuel and the freestream product gas. Following this, the prandtl number can be calculated as per its definition:

(6)

(7)

Next, a second-order Runge-Kutta method is employed to approximate the addition of mass flow rate to the boundary layer from entrainment of the freestream gas in the boundary layer. The finer details of the method will be spared, but the vague framework of the method is that a guess will be made at the change in boundary layer mass flow, after which a new guess will be made based on the first guess, and then the final guess is approximated as the average of the two.

(8)

(9)

…noting again that and are treated as finite, scalar values for this method, and that equation 9 has the output from equation 8 as a parameter. The actual delta-mass flow is then approximated as an average of the two guesses.

(10)

Next, the change in coolant mass flow per unit circumference due to the changing diameter of the nozzle across axial distance - i.e. the decreasing/increasing circumference - must be calculated. This can be approximated using the following equation:

(11)

…with again being treated as a finite delta-D. The change in the total boundary layer mass flow rate can then be approximated by simply adding the “delta-MC” terms calculated above, like so:

(12)

…and the change in coolant (not total boundary layer) mass flow rate per unit circumference can be approximated by neglecting the free stream gas entrainment:

(13)

Next, the recovery temperature, or the temperature of the freestream gas after contacting the boundary layer, must be calculated. The recovery temperature is simply a function of the stagnation temperature (assumed to be essentially equal to chamber temperature as per the chamber modeling document[D]), Prandtl number, and the static temperature as calculated throughout the nozzle in the nozzle modeling document[B].

(14)

Finally, the linearized change in adiabatic wall temperature can be approximated using the provided equation:

(15)

Then, approximating the adiabatic wall temperature at the current step simply becomes:

(16)

For the purposes of determining heat flux, the heat transfer coefficient can be calculated as per the thermal loads document[A]. This process then restarts, with the calculated parameters in equations 3-16 becoming the initial conditions for the next step. The process is repeated for each step until the full length of the nozzle has been covered. Note that after the recovery temperature, wall temperature, static pressure[D], etc. are calculated at each point, the fluid properties of the coolant and freestream gas must be re-evaluated - again, preferably using Cantera.

## Single Differential Modeling of Wall-Gas Temperature

The previous purely numerical method can be used to yield marginally more conservative (i.e. higher temperature) estimates of an expected wall temperature, another method is provided by Grissom which appears to yield curves which more closely reflect the effects of changing recovery temperature throughout the nozzle. This method involves a single differential equation describing the behavior of the cooling effectiveness . The cooling effectiveness is defined simply as…

(17)

…or more simply put, the ratio of the difference between the recovery temperature and the wall temperature to the difference between the recovery temperature and the injection temperature . So, when , the wall temperature is equal to the injection temperature (i.e. no heat flow from the freestream into the wall), and when the wall temperature is equal to the recovery temperature (i.e. no cooling).

This method is centered around the following differential equation describing as a function of contour distance :

(18)

The only difference in the initial conditions and nomenclature are defined as follows:

Initial film cooling effectiveness:

General correction factor:

Parameters such as , , , , and can be calculated in the same manner as specified in the above section. is calculated as a finite value in the same manner, using equation 2. Recovery temperature is again calculated using equation 14.

Assuming an initial cooling effectiveness of , equation 18 is then used to calculate some finite at each point of axial distance starting at . The cooling effectiveness at each point can then be determined using a similar method as equation 16, shown below in equation 19.

(19)

Once discrete values of have been obtained, the wall temperature can then be backed out simply by using the definition of boundary layer cooling effectiveness, equation 17.

(20)

This method is somewhat computationally simpler to implement, and yields a curve which more closely resembles the experimental data shown in TR-91-1[1], but can be more optimistic in its predictions of the effectiveness of film cooling, albeit to a very small degree. As with the expanded method, the coolant ideal gas properties such as density and viscosity need to be recalculated at the calculated static pressures and wall temperatures/recovery temperatures.

Comparisons of these two methods indicate that the predicted wall temperature does not differ significantly until the flow reaches the expanding section of the nozzle, after which the temperature predicted by the single-equation method decreases very slightly below that of the expanded method. Determining which method is more accurate to experimental results, however, requires testing of the rocket engine itself for comparison to theoretical data.

## 

## References and Sources

[1] [Grissom BL Cooling](https://drive.google.com/file/d/1sblwUi_BpPOf-dg0ytU-u0OWHGKqjmZC/view?usp=sharing)

## Related Documentation

[A] [Thermal Considerations and Loads](https://docs.google.com/document/d/1homPFQnGbIEBKH1nYHeWbC9jdXok5Kiczb4tmeqo9O4/edit#)

[B] [Nozzle Flow Modeling](https://docs.google.com/document/d/16WyGRw5Ikb9o1UvMvrIiOay80P_ZhJnjGX2SzdEGsrk/edit?usp=sharing)

[C] [Cantera Quick Reference](https://docs.google.com/document/d/1qyf48IUYjvoWwlnIdurIilh9O9bYv-0zfA_flUvgOP8/edit?usp=sharing)

[D] [Chamber Flow Modeling](https://docs.google.com/document/d/1jLYpAgBIF4DAZvxFLMegq2dOw-f8xS7uXNU71RqToM4/edit#heading=h.bu9knfbvmhgs)