Developing a Workflow to Estimate Heat Transfer for High Altitude Flight

Anish Agrawal
Jaden Hernandez
Shreesh Nalatwad
Noah Smith
William Stupart

School of Aeronautics and Astronautics
Purdue University
May 2, 2025

Abstract

This project presents a MATLAB-based thermal analysis tool designed to estimate the insulation thickness required within the airframe of a sounding rocket. During early design phases, it is crucial to obtain reliable thermal protection estimates without relying on expensive and time-intensive computational fluid dynamics (CFD) simulations. This tool provides a fast, low-cost alternative by using simplified heat transfer models that deliver sufficiently accurate results to inform preliminary thermal protection system (TPS) design. These estimates can then be integrated into broader vehicle sizing models. The tool is especially valuable for student or low-budget rocketry programs, where access to high-fidelity modeling resources may be limited. By inputting rocket geometry, motor performance, material properties, and the desired internal temperature, the tool outputs the minimum insulation thickness needed to maintain that target. While not a replacement for CFD in later stages, this tool offers a powerful capability for early-stage TPS development.

Team Member Contributions

Anish Agrawal: Ansys Transient Thermal Simulation, wrote section 3.2
Jaden Hernandez: Created heat transfer model in MATLAB, wrote sections 1 and 2
Shreesh Nalatwad: Incorporated real-world data to validate simulation, wrote section 3.1
Noah Smith: Cited references, wrote conclusion
William Stupart: Created the dynamics model in MATLAB, wrote section 3.3 and abstract

1 Introduction

Sounding rockets are a renowned medium for conducting experiments and taking measurements in the atmosphere that have been used by researchers for decades. In recent times, there has not only been an influx of university organizations designing sounding rockets as hobby projects, but also an interest from researchers to study microgravity effects that are typically observed in a high-altitude rocket. One facet of rocket design that may prove to be complicated, especially for a student rocketry team, is insulation thickness to thermally protect payloads and avionic systems as the rocket experiences changes in altitude and flow regime.

The intent of this project is to create a simplified workflow for evaluating insulation measures in various different flight regimes, including support for subsonic, supersonic, and high-altitude flight. By providing rough estimates of aerodynamic heating characteristics without the need for long, transient computational fluid dynamics simulations, rocket design teams can effectively accelerate their preliminary design iterations with a basic understanding of the insulation necessary for their mission parameters. Consequently, as student rocketry becomes more accessible through simplified workflows like this, there will be more opportunities for these university rocketry teams to collaborate with researchers to support further microgravity research and prevent similar research areas from stagnating.

2 Workflow Development

The simplified workflow that will be used to estimate heating characteristics was developed in MATLAB by using a large number of heat transfer and compressible flow relations, and features three cases for simulation: case 1, where the user can analyze the predicted trajectory, thermal properties of air at varying altitudes, and heating of the airframe over time; case 2, where the user can evaluate the effectiveness of a provided insulator via an internal temperature over time plot; and case 3, where the user can find the minimum estimated insulation thickness necessary to maintain a desired internal temperature provided as an input. In case 2 and 3, all of the information from case 1 will be provided in addition to the information provided by the unique cases.

2.1 General Methodology

In summary, while the developed MATLAB model is a combination of functions, the script accepts the following inputs, then provides the following outputs depending on the case.

Table 1: Inputs and outputs of MATLAB model

Case	Input	Output
1	Airframe Thermal Conductivity (W/m-K)	Trajectory of Rocket Plot
1	Airframe Density (kg/m^3)	Geometry of Rocket Airframe Plot
1	Airframe Specific Heat (J/kg-K)	Velocity of Rocket Plot
1	Drag Coefficient	Air Thermal Conductivity vs. Altitude Plot
1	Dry Mass (kg)	Air Prandtl Number vs. Altitude Plot
1	Propellant Mass (kg)	Air Specific Heat vs. Altitude Plot
1	Average Thrust (N)	Air Temperature vs. Altitude Plot
1	Burn Time (s)	Heat Flux Onto Airframe vs. Time Plot
1	Airframe Outer Diameter (m)	Airframe Temperature vs. Time Plot

1	Airframe Thickness (m)	Maximum Biot Number
1	Airframe Length (m)	
2	Insulation Thermal Conductivity (W/m-K)	*Case 1 Outputs*
2	Insulation Density (kg/m^3)	Insulation Temperature Plot
2	Insulation Specific Heat (J/kg-K)	
2	Insulation Thickness (m)	
3	Desired Internal Temperature (°C)	Temperature vs. Thickness Plot *+ Case 1 Outputs*
all	Simulation Case	

This table of inputs and outputs is intended to act as a comprehensive list of parameters handled by the function, not necessarily to imply that one input singlehandedly controls the output next to it. The inputs are provided by the user in a simple graphical user interface (GUI) that appears when the user runs the program. From there, the program follows a general approach depicted in Fig. 1 to obtain the results for its given simulation case.

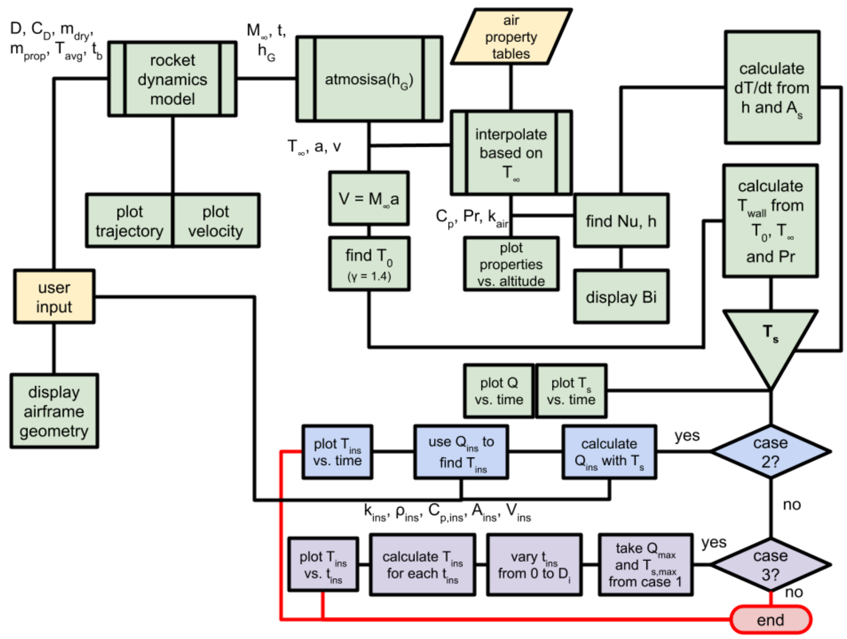


Figure 1: MATLAB script flowchart

2.2 Relevant Equations

Many equations were used to simulate heating, generally revolving around Newton's law of cooling, Fourier's heat conduction law, the Nusselt number, and isentropic flow relations. The primary assumptions made for this simplified workflow are as follows: the airframe is approximated as a cylindrical, lumped body. Perfect thermal contact exists between the airframe's inner surface and the insulation when applicable. The heating process is in quasi-equilibrium such that we can approximate each property with steady state expressions. The impact of shocks is ignored since they are adiabatic. All heat transfer is assumed to be one-dimensional, with the airframe having a uniform temperature and the intermediate temperature of the insulation being considered irrelevant. Radiation is considered negligible.

2.2.1 Case 1 Equations

The driving factor that determines most other parameters in the simulation are the dynamics of the rocket, which are calculated in a function separate from the heat transfer calculations. That function is what provides the range of Mach numbers and altitudes as a function of time, based on the airframe's cross-sectional diameter, coefficient of drag, dry mass, propellant mass, average thrust, and burn time. At any given time step n, the altitude is simply:

$$h_{G,n+1} = h_{G,n} + V_{\infty,n} dt \tag{1}$$

where h_G is the geometric altitude, V_∞ is the rocket velocity, and dt is about 1/10000 of a second. The velocity at this time step was determined as:

$$V_{\infty,n+1} = V_{\infty,n} + \frac{T_{avg} - (m_{dry} + m_{prop,n})g - \frac{1}{2}\rho_\infty V_{\infty,n}^2 A_s C_D}{m_{dry} + m_{prop,n}} \tag{2}$$

such that T_{avg} is the average thrust, m_{dry} is the dry mass of the rocket, m_{prop} is the propellant mass, g is the gravitational constant, ρ_∞ is the density of air, A_s is the airframe surface area, and C_D is the coefficient of drag of the airframe. The flight time, $t_{?}$, is simply the total time it takes before the altitude is at or below 0.

There are two primary heating components necessary to determine the heat transfer and airframe surface temperature in case 1: the heat transfer from stagnation at the top of the airframe, and the heat transfer from forced convection of the air contacting the airframe. Regarding the former, California Institute of Technology's Douglas Mackay discovered in his 1953 thesis, "Boundary Layer Temperature Recovery Factor on a Cone at Nominal Mach Number Six" that while

$$T_{wall} = T_\infty + r(T_0 - T_\infty) \tag{3}$$

the recovery factor, r, could actually be approximated as $\sqrt{Pr}$, the square root of the air's Prandtl number[2]. In Eq. (3), T_{wall} is the effective airframe surface temperature after compressibility effects, T_∞ is the surrounding air temperature, and T_0 is the stagnation temperature derived from isentropic flow relations:

$$T_0 = T_\infty\left(1 + \frac{\gamma-1}{2}M_\infty^2\right) \tag{4}$$

in which we assume $y = 1.4$ for air while M_∞ is determined by the dynamics model.

From Žukauskas' 1972 article "Heat Transfer from Tubes in Crossflow", the following relation for the Nusselt number is obtained if the rocket airframe is assumed to be perfectly cylindrical:

$$Nu = 0.027 Re^{0.805} Pr^{1/3} \tag{5}$$

[1] where Nu is the Nusselt number and Re is the Reynolds number of the flow. Generally, for a cylinder, it is well known the heat transfer coefficient is related to the Nusselt number, thermal conductivity, and characteristic length:

$$h = \frac{(Nu)k_{air}}{D_o} \tag{6}$$

such that h is the heat transfer coefficient, k_{air} is the thermal conductivity of the air, and D_o is the airframe's outer diameter. By obtaining h, lumped system analysis can now be used in most analyses, especially if the airframe is thin (under ~0.1 m thick) since even in hypersonic flight with a 0.1 m thick airframe, the maximum Biot number, Bi, yielded is around 0.09. Bi can be expressed as:

$$Bi = \frac{hL_c}{k} \tag{7}$$

where L_c is the characteristic length and k is the thermal conductivity of the airframe. Since Bi is practically always less than 0.1, we can assume the airframe to be a lumped body. The temperature change contribution from the forced convection is defined as:

$$\frac{dT_s}{dt} = \frac{hA_s}{\rho \upsilon c_p}\left(T_\infty - T_s\right) \tag{8}$$

so that $\frac{dT_s}{dt}$ is the change in surface temperature with respect to time, A_s is the airframe surface area, ρ is the density of airframe material, υ is the volume of airframe, c_p is the specific heat of airframe material, and T_s is the outer surface temperature. In MATLAB, this is discretized into time steps, where each iteration can be denoted as subscript n. In this sense, the predicted temperature at the next time step when combining T_{wall} and dT_s is:

$$T_{s,n+1} = T_{wall,n} + \frac{h_n A_s}{\rho \upsilon c_p}\left(T_{\infty,n} - T_{s,n}\right)dt \tag{9}$$

where h, T_∞, T_s, and T_{wall} are varying with each step n and dt is equal to 1/10000 of a second. At the new time step, the airframe heat transfer, Q_a, is calculated from Newton's law of cooling:

$$Q_{a,n} = h_n A_s (T_{s,n} - T_{\infty,n}) \tag{10}$$

2.2.2 Case 2 Equations

For case 2, T_s of the airframe is assumed to be independent of position due to the lumped system analysis approximation. Therefore, both the outer and inner surface temperatures are the same. This means the surrounding temperature in the conduction problem is T_s. First, the heat transfer into the insulation, Q_{ins}, is calculated at a particular time step n as:

$$Q_{ins,n} = \frac{k_{ins} A_{ins}}{t_{ins}} (T_{s,n} - T_{ins,n})dt \tag{11}$$

with k_{ins} being the insulation's thermal conductivity, A_{ins} being the surface area of the insulation, t_{ins} being the insulation thickness, and $T_{ins,n}$ being the internal insulation temperature. With Q_{ins} obtained, by using the well known expression:

$$Q_{ins} = mC_{p,ins} \Delta T_{ins} \tag{12}$$

where m is the insulation mass, $C_{p,ins}$ is the insulation's specific heat, and ΔT_{ins} is the change in insulation temperature, ΔT_{ins} can be isolated to produce a similar approach to Eq. (9)

$$\Delta T_{ins} = \frac{Q_{ins}}{mC_{p,ins}} = \frac{Q_{ins}}{\rho_{ins} \upsilon_{ins} C_{p,ins}} \tag{13}$$

In Eq. (13), ρ_{ins} is the insulation material's density while υ_{ins} is the insulation volume. Subsequently, then, the predicted insulation temperature at the next time step is trivial:

$$T_{ins,n+1} = T_{ins,n} + \Delta T_{ins,n} \quad (14)$$

2.2.3 Case 3 Equations

For case 3. where the insulation thickness, t_{ins} is unknown, an array of possible values is created such that t_{ins} can vary from 0 to D_i, the inner diameter of the airframe. The maximum heat transfer and maximum airframe temperature from case 1, denoted as $Q_{a,max}$ and $T_{s,max}$ respectively are utilized to find the maximum insulation temperature, $T_{ins,max}$, at each thickness. By rearranging Eq. (11), the maximum internal temperature is:

$$T_{ins,max} = T_{s,max} - \frac{Q_{a,max} t_{ins}(0 \rightarrow D_i) dt}{k_{ins} A_{ins}} \quad (15)$$

Obviously, since perfect thermal contact is assumed which eliminates convection, there is no critical radius of insulation which means that the insulation's efficiency will improve as its thickness increases. However, it is usually in the best interest of the user to minimize the thickness of the insulation to save weight and space, meaning that when $T_{ins,max}$ is plotted as a function of t_{ins}, the user should look for the minimum t_{ins} with which $T_{ins,max}$ intersects with their desired temperature.

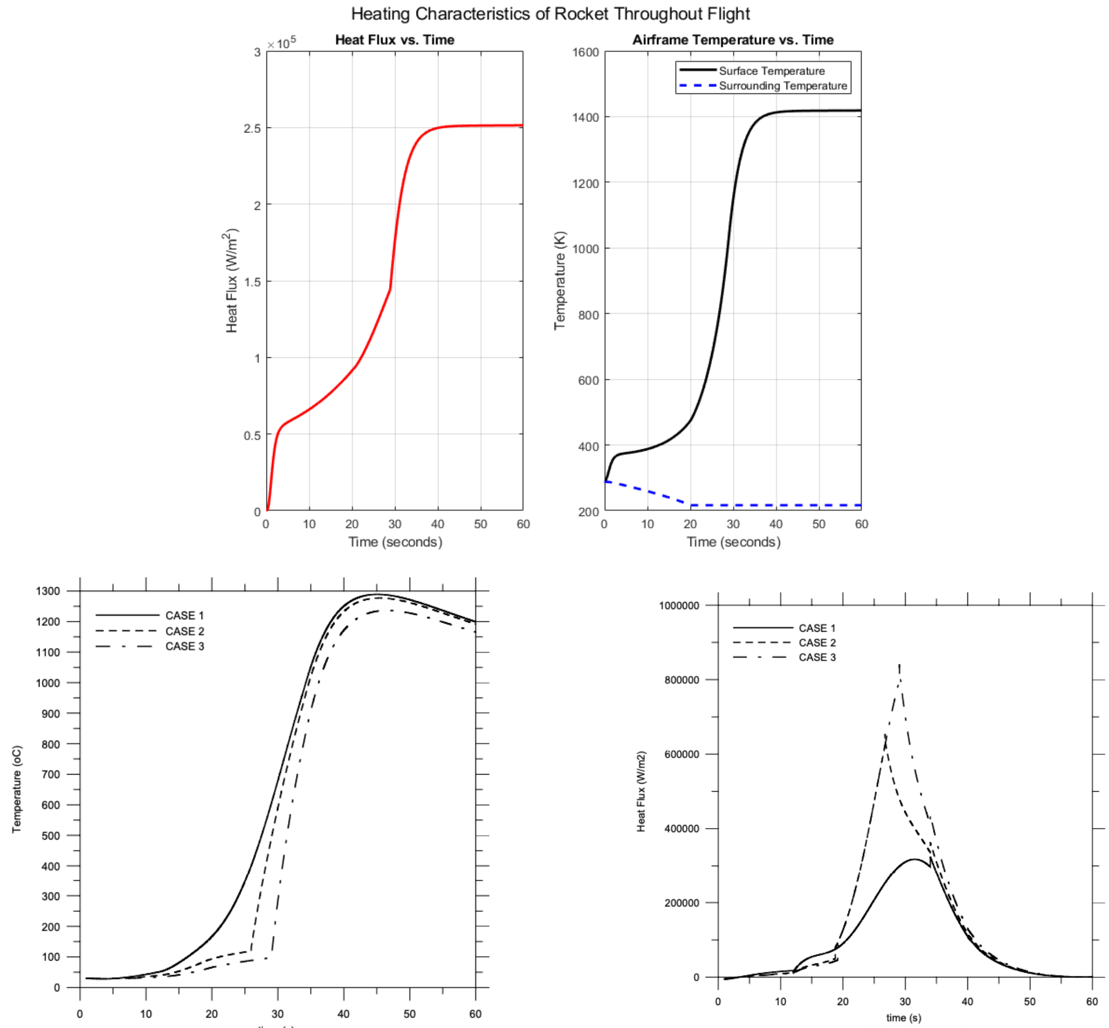
3 Results and Analysis

3.1 Case 1 Results, Convection Verification

Case 1 is designed to accept user defined flight parameters to simulate the aerodynamic heating experienced by the airframe as it follows its predicted flight path. The user can analyze temperature and heat flux trends over time to help select thermal insulation and verify that the airframe remains within safe temperature requirements.

To validate our thermal simulation code for the rocket airframe, we examine a real world case study based on the Brazilian VSB-30 Sounding Rocket launched in October 2004. A 2005 study analyzed aerodynamic heating on the rocket fins during hypersonic flight using convective heat flux calculations and analyzing the effects of ablative material. For our case, we are only relying on the base case that does not use any ablative material as to provide the most accurate comparison to our simulation data.

Using our thermal simulation tool, the input parameters include airframe thermal properties, rocket geometry, dry & propellant mass, average thrust, and other important properties to provide the most accurate results. Inputting the relevant characteristics for the VSB-30, here are the simulation results compared with the experimental data using a thermocouple on one of its fins with simulation above and experimental below.



(a) Temperature (b) Heat Flux

"Case 1" in the graph above refers to the fin with no ablative material, which is most relevant for comparison

Figure 2: Comparison of MATLAB analysis and experimental VSB-30 heating data[3]

The temperature profiles from the simulation align closely with the experimental data, with peak temperature at approximately 1400 K, compared to the 1500 K recorded by the thermocouples. Similarly, the heat flux peaks at around W/m^2, and the experimental data peaks around 300,000 W/m^2. One notable difference is in the decay behavior where the experimental heat flux drops to 0 after 60 seconds while the simulation maintains a constant heat pattern until it drops to 0 only at 100 seconds. This is most likely due to the difficulty replicating the flight model and path of the VSB-30, which is a two stage rocket, while our simulation assumes it is a single-stage. Additionally, experimental data comes from thermocouples placed on the surface of the fins, which experience different aerodynamic heating conditions compared to a typical airframe. Another source of error could be due to the measurement lag and calibration issues that could cause the data to vary. Despite these issues, we still believe the comparison is valid due to comparable simulated flight path and environment, and can provide confidence in the code's ability to capture the transient thermal behavior.

3.2 Case 2 Results, Conduction Verification

Case 2 uses user-defined insulation properties such as thermal conductivity and specific heat to generate a temperature vs. time graph at the inner surface. This helps evaluate how effective the insulation is under different flight conditions and payload needs.

With convection between the air and airframe verified, the next step is to validate conduction between the airframe and the insulation layer. This is done using Ansys' Transient Thermal module, which simulates unsteady heat transfer over time based on user-defined material properties and boundary conditions. The outer airframe is modeled as a 1 cm thick steel shell subjected to convection, with time-dependent convective heat transfer coefficients calculated from Reynolds and Nusselt number correlations using flight velocity and altitude data. Heat is then conducted inward to a 10 cm thick layer of rigid polyurethane foam. All geometry and material properties were taken from the Ansys materials library and used as inputs in the model to generate a temperature vs. time graph for the inner surface of the insulation.

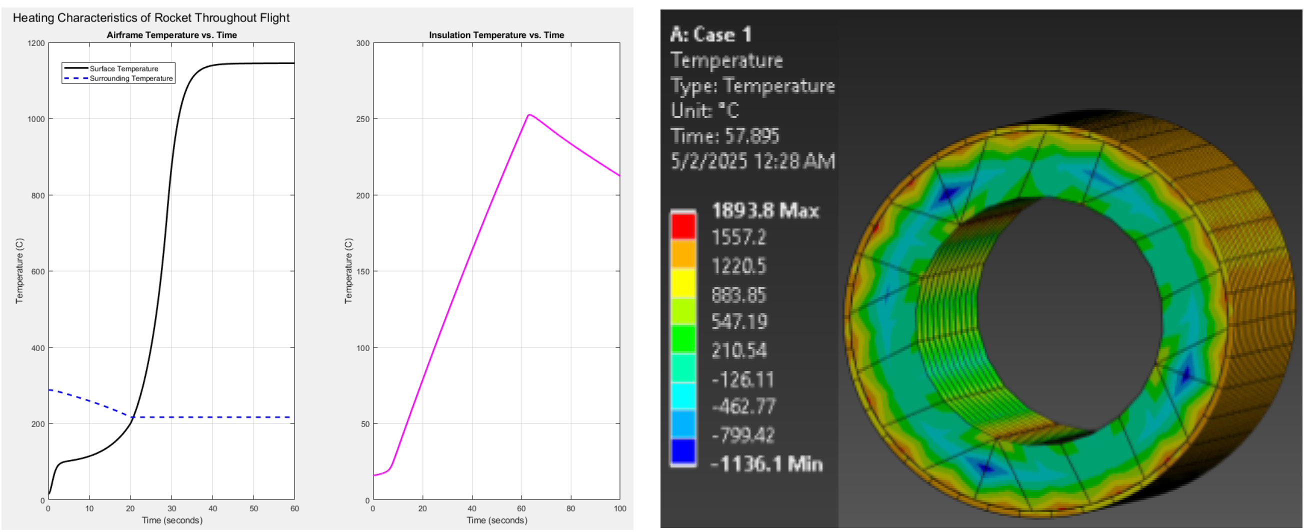


Figure 3: Comparison of MATLAB analysis and Ansys Transient Thermal Module results

Comparing the results, we observe that the outer airframe temperature predicted by the analytical model closely matches the Ansys simulation result, with values of approximately 1145.15°C and 1220.5°C, respectively, at around 57.9 seconds. This corresponds to a percent error of only 6.18%. Similarly, the inner wall temperatures are 233.7 °C for the model and 210.54°C for the simulation, yielding a percent error of 9.91%. These discrepancies could be further reduced by refining the mesh size in Ansys or coupling the simulation with Ansys Fluent to more accurately resolve the airflow and convection dynamics, rather than relying on empirical film coefficient estimates. Additional sources of error may include assumptions of uniform material properties, simplified boundary conditions, or time step resolution. Addressing these factors could enhance accuracy. Given the simplifications in the analytical model, these percent errors are reasonable and indicate good agreement for a first-order thermal validation.

Overall, the close alignment between the model and simulation results supports the validity of the conduction model and its usefulness for evaluating insulation performance under realistic flight conditions.

3.3 Case 3 Results

Now that we have determined that our model outputs reasonable results, we can put it to work by rearranging the parameters to solve for the minimum thickness of insulation necessary to maintain the internal temperature of the rocket at a specified value. For example, if we want to keep the avionics at a stable 20℃, 20℃ can be inputted into the model as the desired internal temperature and with the material properties of the insulating material, and the minimum thickness of the insulated material can be calculated to achieve the desired internal temperature.

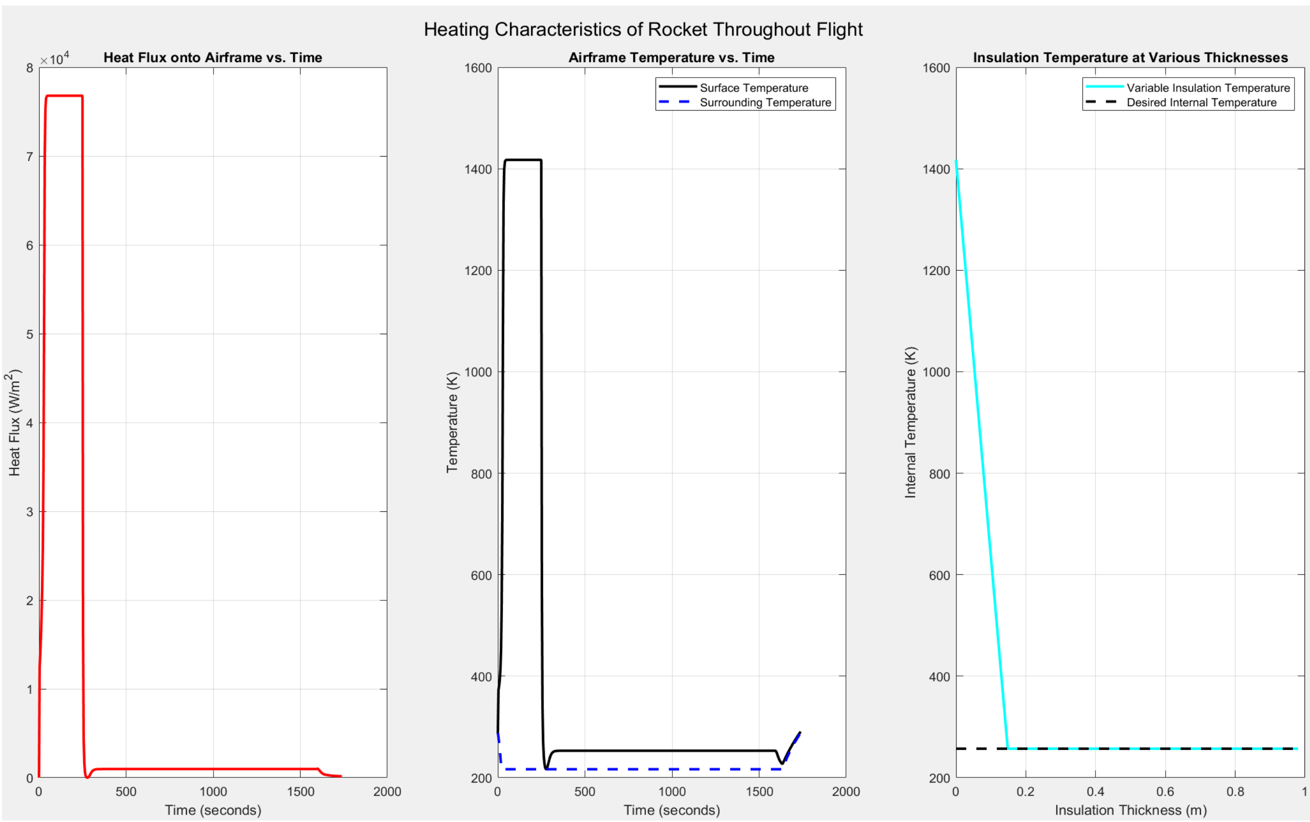


Figure 4: Case 3 analysis outputs

The additional graph on the right hand side in Fig. 4, is of inertial temperature vs. insulation thickness. The light blue line shows how the internal temperature changes as a function of insulation thickness. The dotted black line is the desired internal temperature. The intersection of these two lines represents the minimum insulation thickness required. Any insulation past this, would not change the internal temperature and would only add mass and volume taken up by the insulation. This represents the design condition that our model outputs.

4 Conclusion

Without committing to a design early, there's no good way for traditional CFD methods to evaluate heating on a rocket. This workflow can rapidly provide a good estimate for that parameter early in development before a design is chosen that could turn out to be suboptimal. More importantly, a case 3 analysis can provide a reasonable estimate of insulation sizing which allows designers to understand how much space can be allocated to avionics and payloads.

Many variables can quickly be entered into the model via the GUI, which helps users see how different designs would affect the insulation needed. However, there is room for improvement in this area. Further development of this model should involve applying it to horizontal flight (jets and rocket planes), exploring more detailed rocket/nose cone geometry, and fin analysis on rockets. This could increase simulation time, but would ultimately provide higher accuracy and applicability to more cases. Further experimental data could also improve model accuracy. Our model similarly makes several assumptions; the quasi-equilibrium assumption, for example, could be discarded in favor of a transient finite difference method.

In conclusion, our MATLAB program provides a simplified way for sounding rockets to rapidly size their insulation to protect sensitive components. The model is verified by real-world analysis similar to cases 1 and 2. These results can make for more efficient development of smaller-scale rocketry for the relevant users.

5 References

[1] A. Žukauskas. Heat Transfer from Tubes in Crossflow. Advances in Heat Transfer, 93 (1972).

[2] D. Mackay. Boundary Layer Temperature Recovery Factor on a Cone at Nominal Mach Number Six. California Institute of Technology, (1953).

[3] J. Mazzoni, J. Filho, H. Machado. Aerodynamic heating on VSB-30 sounding rocket. 18th International Congress of Mechanical Engineering (2005).