

A STREAMLINED APPROACH TO SPATIAL MAPPING OF COMPLEX 3D THERMAL BOUNDARY CONDITION DATA

Robert F. Hawkins
(NASA/Marshall Space Flight Center)

ABSTRACT

Spatial mapping of 1D, 2D, and particularly 3D thermal boundary condition data is essential for accurate thermal simulation of spacecraft and launch vehicles. This information typically is available in simple ASCII file format but can be problematic to use for thermal simulations. Difficulties that can be encountered include the ability to directly use the available file format, transformation of the data into the desired reference coordinate system in 3D space, and accurate interpolation of the data onto the thermal calculation domain.

This paper presents a streamlined approach to spatial mapping of complex unsymmetrical 3D cold wall heat flux data using the fields' capability in Simcenter3D. A recent example of 3D thermal plume simulation at MSFC will be employed to illustrate the use of text, csv, and Excel files, specifying coordinate systems for transformation of data in 3D space, and use of the various interpolation schemes available for spatial mapping. Spatial boundary condition verification is also very important, and the spacecraft thermal analyst needs effective visualization tools to develop confidence in the boundary condition definitions. Visualization of the spacecraft plume raw field data, the calculated field data, and the interpolated field data onto the thermal calculation domain will be discussed, with a particular focus on real-time viewing of the interpolated data for available interpolation schemes.

NOMENCLATURE, ACRONYMS, ABBREVIATIONS

P	Pressure
T_{aw}	Adiabatic Wall Temperature
T_{rec}	Recovery Temperature
q_r	Incident Radiative Plume Flux
q_{cw}	Cold Wall Convective Plume Flux
T_{am}	Ambient Temperature
T_{cw}	Cold Wall Temperature

INTRODUCTION

The objective of this paper is to outline the modeling techniques used with the integrated thermal analysis of a terrestrial lunar lander demonstrator. Specifically, the techniques outlined herein focus on spatial mapping of convective and radiative exhaust plume results from a computational fluid dynamics (CFD) analysis. Two cases were investigated, however, the modeling techniques that were utilized translate to both cases seamlessly. Those two cases were as follows:

- Lunar lander demonstrator firing on-pad (worst case).
- Lunar lander demonstrator hovering at an elevated position above the launch pad.

For the purposes of information control, this paper will focus on the modeling techniques only, using altered or dimensionless data where necessary.

BACKGROUND

The thermal model examined in this paper is of the lunar lander demonstrator known as XL-1T (terrestrial), born from a collaborative effort between Masten Space Systems (MSS) and NASA, under NASA's Lunar CATALYST (also known as Lunar Cargo Transportation and Landing by Soft Touchdown) initiative. The lander is a reusable vertical takeoff/vertical landing (VTVL) test bed which is controlled by four throttleable main engines utilizing green hypergolic propellants.

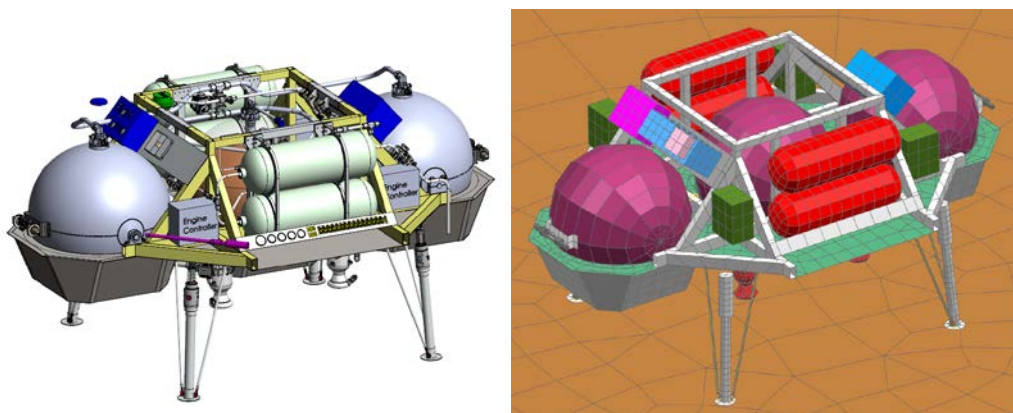


Figure 1. Masten XL-1T lunar lander demonstrator with associated thermal model.

The positioning of the main engines of the vehicle creates a unique environment wherein the base-flow physics creates a fountain plume that impinges directly on the base of the vehicle, greatly increasing convective plume loads. The aforementioned two plume study cases differ in that the fountain plume reacts differently to the adjacent nozzle plumes from the main engines. This difference is primarily attributed to the Coanda effect. The Coanda effect is the tendency of a fluid jet to be attracted to an adjacent flat or curved surface. While the vehicle is on-pad, the fountain plume oscillates between nozzle plumes, never fully attaching to any one nozzle

plume and thus moves about the center of the vehicle base. While the vehicle is elevated, the fountain plume weakens enough to attach to two of the nozzle plumes, reducing base heating.¹

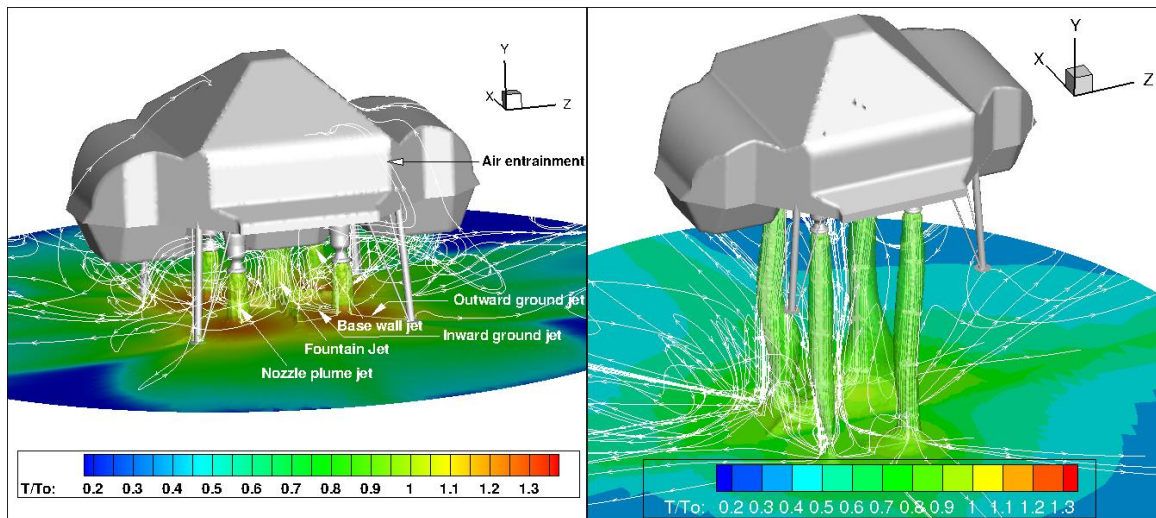


Figure 2. On-pad (left) and elevated (right) plume flow fields.

PLUME ENVIRONMENT CHARACTERIZATION

The asymmetric geometry of XL-1T combined with the complex plume interactions creates a difficult plume environment to map to the thermal model. As mentioned previously, the Coanda effect accounts for an unpredictable and sporadic plume loads. In this instance, the four adjacent nozzle plumes caused the fountain plume to oscillate across the vehicle base in the on-pad case and attach to two of the nozzle plumes during the elevated case. This phenomenon makes mapping plume flux data difficult in a spatial sense due to varying flux values at any given point on the vehicle surface.

The plume data obtained from the CFD analysis was broken down by body points according to Cartesian coordinates on the vehicle surface. With each location point the following data was provided:

- Pressure (P)
- Adiabatic Wall Temperature (T_{aw}), or Recovery Temperature (T_{rec})
- Incident Radiative Plume Flux (q_r)
- Cold Wall Convective Plume Flux (q_{cw})
- Ambient Temperature (T_{am}), or Cold Wall Temperature (T_{cw})

Radiative flux values were incident values calculated using estimated optical properties, so the values could be applied directly to the vehicle surface using the corresponding points. However, convective flux values and accompanying convective heat transfer coefficients change depending on the vehicle temperature, requiring constant recalculation throughout the analysis.

For these calculations the convective heat transfer formula can be written as the following:

$$q_c = h_c \cdot (T_{rec} - T_{surface})$$

Equation 1. Convective heat transfer equation for the lander surface.

Where h_c is the convective heat transfer coefficient and $T_{surface}$ is the temperature of the vehicle surface.

To account for the changing convective heat transfer, the same equation can be written with respect to the cold wall flux:

$$q_{cw} = h_c \cdot (T_{rec} - T_{cw})$$

Equation 2. Convective heat transfer equation as a function of cold wall heat flux.

Where T_{cw} is the ambient temperature of the vehicle.

Combining the two equations yields a relationship which calculates the convective flux while accounting for the changes in the convective coefficient:

$$\frac{q_c}{q_{cw}} = \frac{T_{rec} - T_{surface}}{T_{rec} - T_{cw}} \text{ or } q_c = q_{cw} \cdot \frac{T_{rec} - T_{surface}}{T_{rec} - T_{cw}}$$

Equation 3. Combined convective flux equation, accounting for changing heat transfer coefficient.

THERMAL MODEL APPLICATION METHOD

A select group of components were chosen for the focus area of plume application in the thermal model. These components were the highest risk/most affected areas and the ultimate goal of the analysis is to provide design inputs and recommendations for plume shielding and general thermal protection for the vehicle. The CFD results were distributed per the groupings shown in Figure 3. There were three tank plume shields (domes), four leg assemblies (with one compression and two tension members each), the plume shield closeout (separating the upper half of the vehicle from the plume), and the four main engines.

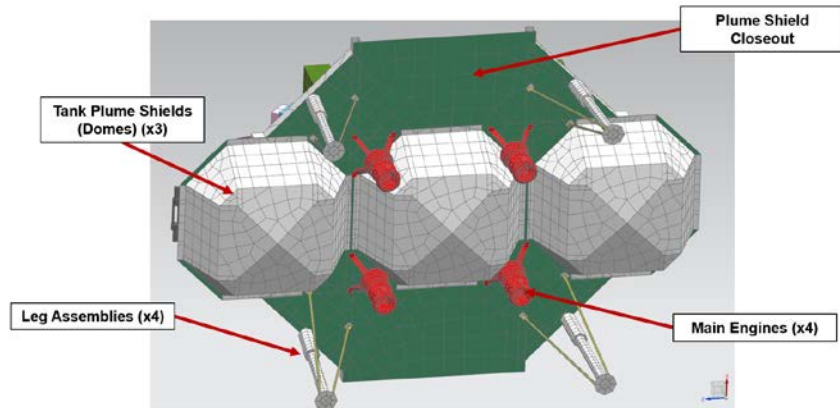


Figure 3. Plume focus areas for thermal model.

The thermal modeling was performed using Siemens Simcenter3D (SC). Simcenter was chosen primarily for the fields capabilities that allowed direct inputs of the .csv raw data from the CFD analysis via embedded Excel files. These fields were separate Cartesian based fields with dependent variables representing q_{cw} , q_r , and T_{rec} from the CFD plume data. Furthermore, the fields were then delimited by associated component geometry of the vehicle (e.g. q_{cw} linked to plume shielding, q_{cw} linked to legs, etc.). Figure 4 shows the direct link between the SC Table Field and the raw data. Due to the density of the received plume data, a Nearest Neighbor interpolation scheme could be chosen from the options to apply the mapping without sacrificing fidelity.

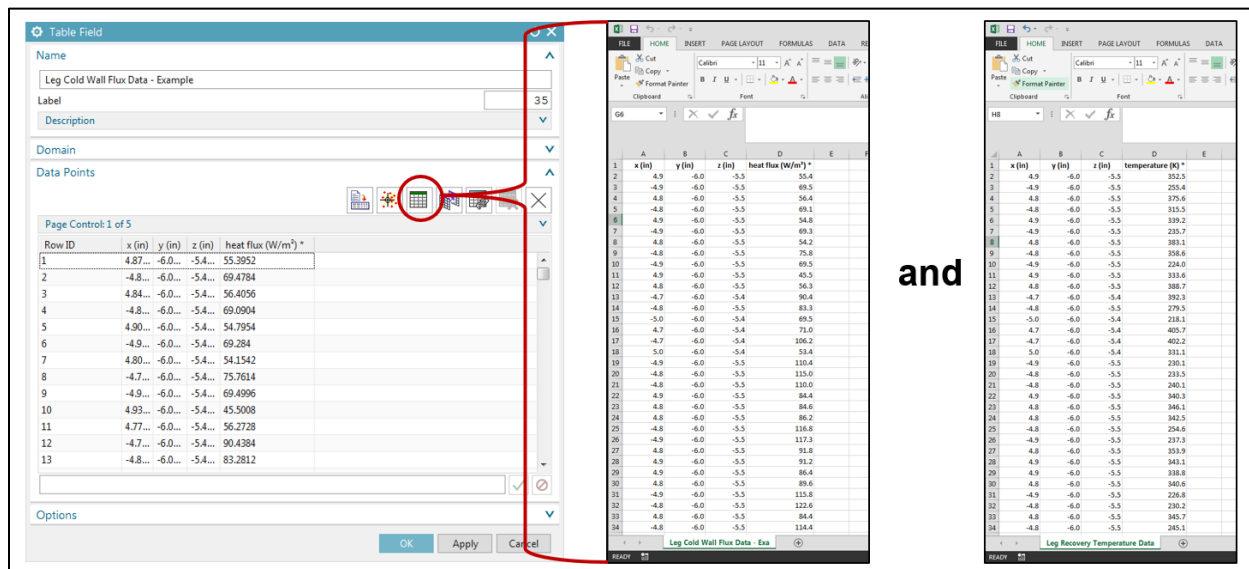
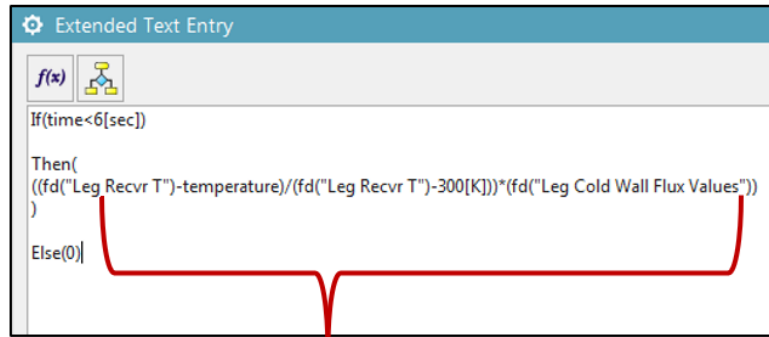


Figure 4. Table Field option showing inputs for convective and radiative plume data.

With the data represented in appropriate fields, loads were created and mapped to applicable geometry in the model. To utilize the preceding equations conditional statements were used in the magnitude field directly to recalculate convective flux values as the vehicle temperature

changes. An If/Then statement is used to tie the transient time stamp to the plume loads. For example; *If(time<'firetime'[sec]) Then('load calculation') Else(0)*. In this statement, the model begins with the plume loads active, and deactivates the loads when the desired time eclipses 'firetime'. To activate the plume loads at another time within the transient analysis an additional qualifier could be added, e.g. *If(4[sec]<time<27[sec])*. In this example, the 'Then' statement would only trigger when the time stamp was between 4 and 27 seconds during the transient analysis. When the qualifying 'If' statement is satisfied, either the convective plume (Figure 6) or radiative plume (Figure 7) loads will be active in the analysis. Using the convective flux relation in Equation 3, the convective plume flux can be calculated as the vehicle temperature changes. To represent this equation, the fd ("field name") command can be used to fill the variable roles of the T_{rec} , T_{cw} and q_{cw} by recalling the fields created from the CFD data. By using the term 'temperature' the temperature of the vehicle's elements can be referenced and by using 300[K], the cold wall temperature can be set. One further note is that when using straight numbers bracketed units must be used, while the field units are set when the fields are created. Finally, the radiative plume loads can be invoked just by recalling the field name and using the time conditional statement to ensure the loads are active at the appropriate times.

Figure 5. Thermal Loads dialog for choosing geometry and using Equation 3 as the Magnitude.



$$q_c = q_{cw} \cdot \frac{T_{rec} - T_{surface}}{T_{rec} - T_{cw}}$$

Figure 6. Extended Text Entry view of thermal load magnitude field, with conditional statements for fire time and field recall statements forming Equation 3.

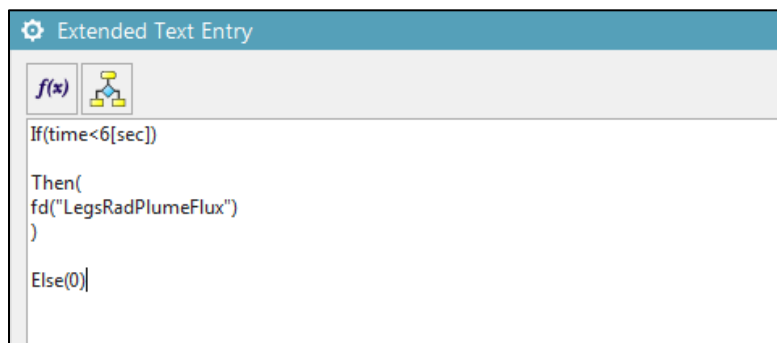


Figure 7. Extended Text Entry view of thermal load magnitude field, with conditional statements for fire time and field recall statement for radiative flux values.

To check the mapping and convective calculations, the plume loads can be plotted via contour plots. This feature provides options to manually input surface temperatures for the resulting contour calculations, or animating across a span of surface temperatures.

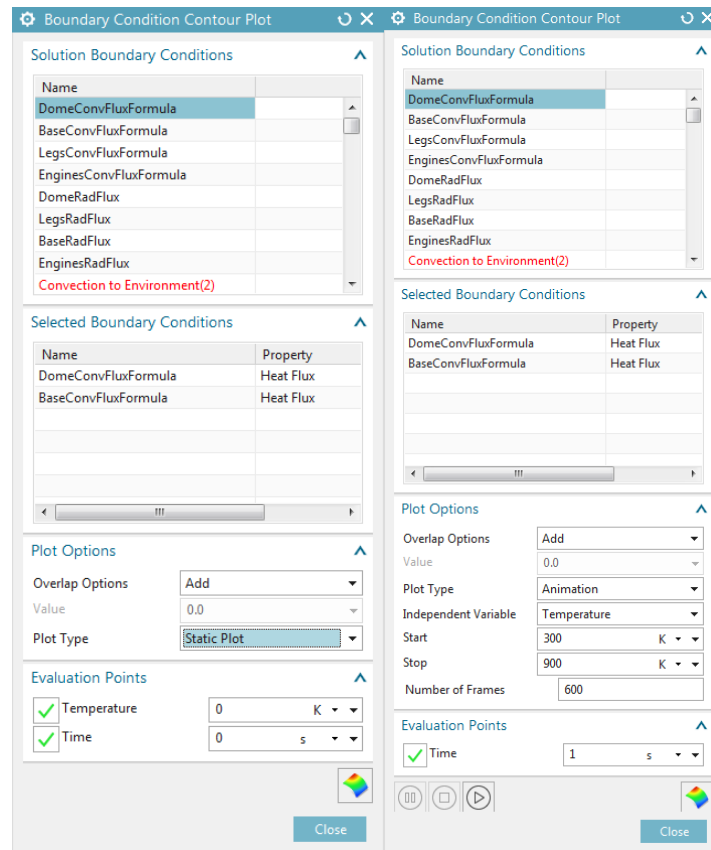


Figure 8. Boundary Condition Plotter showing static plot via Temperature/Time and animated plot across temperatures with given time.

CONCLUSIONS

In conclusion, the plume loads mapped closely relative to CFD results and integrated thermal analysis results tracked as expected. Vehicle temperatures appeared to respond appropriately and soak back periods could be determined from the same analysis. The application of plume loads was sped significantly by utilizing .csv raw data without data manipulation or external script routines. By utilizing the raw data on a point by point basis, along with real time calculations using multiple variables provides a more accurate depiction of the plume environment without sacrificing fidelity. The tradeoff for this method seems to come in the form of extended run times, however. There may be more optimization that could be utilized in the methods, but as of this writing, analysis times were prohibitive to parametric runs. It may be possible to link different analysis cases together to curtail the run times, by quarantining the plume calculations to a fire time only analysis, but this hypothesis has not yet been tested. All in all, this modeling method simplifies and expedites the process of mapping plume, but the increases due to real time calculations could ultimately encroach on the added time savings.

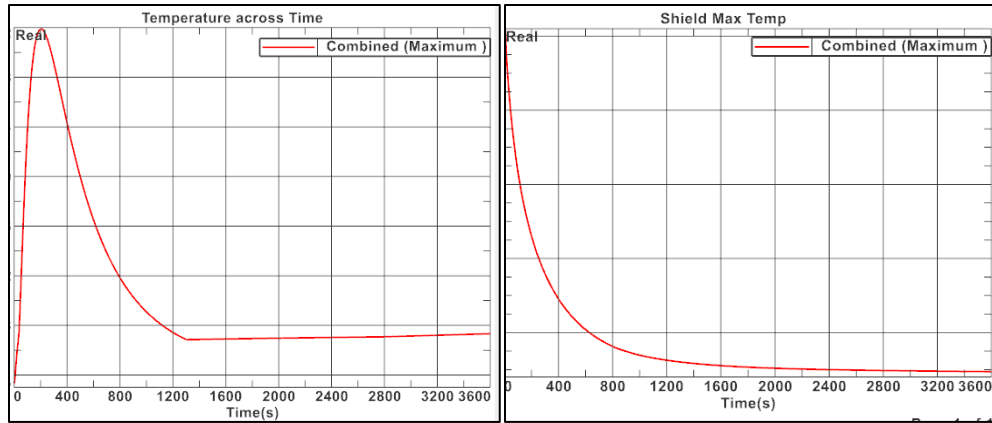


Figure 9. Temperature versus time plots for plume affected geometries.

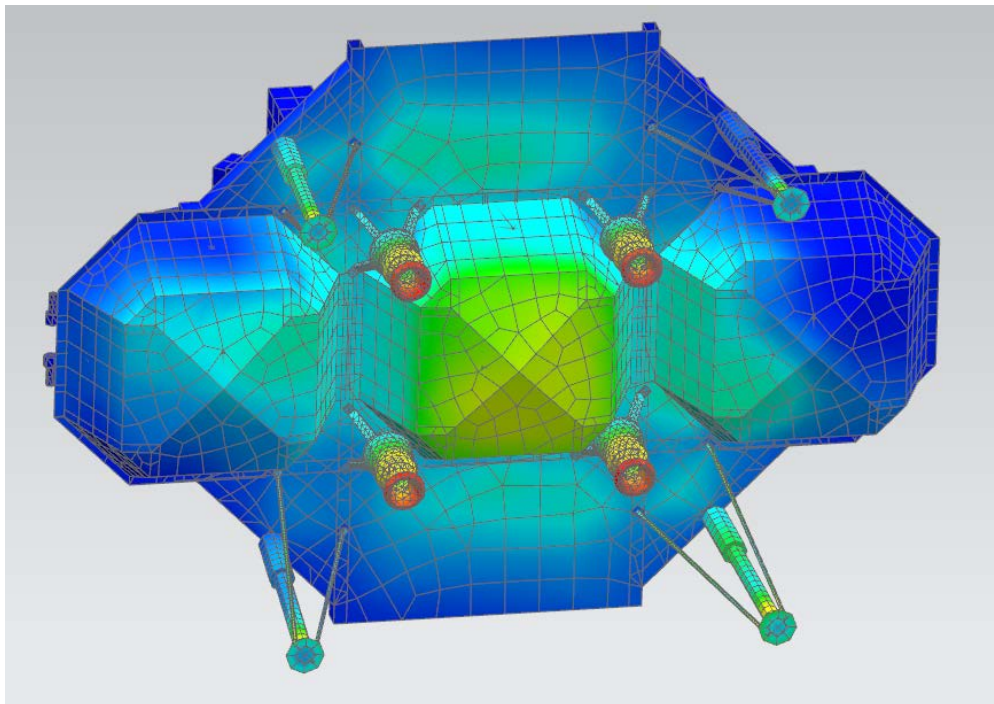


Figure 10. Temperature contour plot showing plume affected areas of lander model.

ACKNOWLEDGEMENTS

- Carl Poplawsky (MAYA Heat Transfer Technologies)
- Jean-Frederic Ruel (MAYA Heat Transfer Technologies)
- Ten-See Wang (NASA/MSFC)
- Kyle Knox (NASA/MSFC)
- Matt Bergman (Masten Space Systems)
- Dan Popok (Jacobs ESSA/MSFC)

CONTACT

Bob Hawkins (NASA/MSFC)
Robert.Hawkins@nasa.gov
(256)961-1763

REFERENCES

1. T.S. Wang, F. Canabal, K.S. Knox, R.F. Hawkins (NASA/MSFC), and M. Bergman (MSS).
“Analysis of Base-Heating Environment During Ground Testing of a Lunar Lander
Demonstrator”. JANNAF 2019