Analysis of heat transfer from a local heat source at cryogenic temperatures

W L. Johnson1, R Balasubramaniam1,2, and R Grotenrath1

1 Glenn Research Center, Cleveland, OH 44135 USA
2 Case Western Reserve University, Cleveland, OH 44106 USA

E-mail: Wesley.L.Johnson@nasa.gov

Abstract. Understanding the dispersion of heat around a cryogenic fluid tank, specifically the interaction between the cryogenic fluid and the tank wall is critical in the analysis of long duration cryogen storage in microgravity. The heat transfer interaction between a cryogenic storage tank and heat sources from external spacecraft structures is also one of the many factors that determine how much heat enters a tank. Recent flight experiments with two-phase fluids have indicated that local concentrations of heat input (also known as “hot spots”) can cause unwanted effects including local boiling. Computational fluid dynamic (CFD) models can provide a detailed assessment of the heat transfer occurring across a cryogenic storage system. However, CFD modeling takes time to construct and run. A simpler approach that can act as initial guidance for later CFD modeling analyzes external “hot spots” as point or finite heat sources. A radial, finite element network or a local direct solution can effectively estimate the heat spread across a cryogenic storage tank by calculating the temperature and heat load as a function of distance from the heat source. This calculation accounts for the convective heat transfer between the cryogenic fluid and storage tank surface. Similar approaches can be used to determine the effectiveness of cooling from a cryocooler as a finite, local heat sink. This approach allows for quick approximations of the thermal map across a cryogenic tank as well as sensitivity analysis under a wide range of design parameters including gravitational fields as implied through natural convection coefficients.

1. Introduction

Natural convection of heat within a cryogenic tank, especially heat generated by concentrated heat loads provides a challenging environment once gravity is removed from the system. Gravity driven natural convection is strong enough to smooth out these point heat load or “hot spots” in the 9.8 m/s² gravitational environment on Earth. However, in reduced gravity the intensity of natural convection is diminished. Therefore, heat removal from hot spots is curtailed. For liquids that are in nearly saturated conditions, it is possible for the liquid in the vicinity of hot spots to be superheated. This can promote the onset of nucleate boiling in the tank.

The liquid superheating may lead to unwanted consequences. The Tank Pressure Control Experiment (TPCE) [1] investigated the boiling process adjacent to small heaters at low heat fluxes in a microgravity environment. Freon 113 was the test fluid and the liquid (at a fill level of 83%) was slightly subcooled. At heat fluxes in the range 0.86 to 1.16 kW/m², it was found that high wall superheats could be sustained for long periods of time without the occurrence of nucleation and boiling. Thus, the liquid adjacent to the heaters would be superheated as well. In one test, a wall superheat of 17.9 C was achieved after 10
min of heating at a heat flux of 0.97 kW/m². Furthermore, the superheating phase was followed by explosive boiling that resulted in a pressure spike of 38\% of the tank pressure at the inception of boiling. For lower heat fluxes (as in cryogenic tanks insulated by MLI), it would take more time for the nucleation of bubbles on the tank wall. The penetration depth of the superheated liquid layer adjacent to the heat load would increase, causing the potential for more intense explosive boiling to occur.

More recently, the Zero Boil-Off Tank Experiment (ZBOT) \cite{2, 3} was performed on the International Space Station to investigate interdependent mechanisms of forced mixing of the liquid, gravity-dependent transport processes in the liquid and vapor, and evaporation and condensation at the liquid/vapor interface. The test fluid used is Perfluoro-n-Pentane (PnP, or C5F12), which is a non-polar volatile refrigerant with a boiling point of 29°C at 1 atm and a near zero contact angle with the transparent acrylic test tank. To emulate self-pressurization of cryogenic tanks, a temperature-controlled vacuum jacket surrounding the test tank as well as heating by strip-heaters mounted on the exterior tank wall was employed. Pressure control was accomplished by a temperature-controlled liquid jet flow directed at the ullage. Some of the experimental findings were: (i) deviations from the classic self-pressurization trends can occur more readily in Space due to the weaker residual natural convection and stronger localized thermal stratification in microgravity as compared to their 1g counterparts, especially if the wall also provides suitable nucleation sites. During strip heating, boiling was initiated at two RTD locations attached to the smooth tank wall along the heated strip. The RTD attachments provided a rough surface that was conducive for bubble nucleation and the absence of a strong natural convection allowed a high $\Delta T_{\text{incipience}}$ of 5-6 degrees (required for nucleate boiling) to be reached at a very moderate heater power of 0.5 W (heat flux of 128 W/m²) (ii) during pressure control tests with a subcooled jet, surprising and sudden evolution and growth of many small bubbles in the tank was observed. The bubbles were initially small, but subsequently grew and coalesced to make larger bubbles. It is suspected that as the tank pressure decreased during mixing with the subcooled jet, the saturation temperature dropped below the temperature of the screened metallic liquid acquisition device (LAD). Both sides of the LAD acted as hot spots during the pressure collapse and caused massive boiling or a cavitation process that was unexpected.

The effect has also been observed in a cryogenic fluid. During the Robotic Refueling Mission #3 (RRM3), evidence from both temperature sensors and Radio Frequency Mass Gauging results suggested that when the cryocooler was turned off, the ullage moved to the cryocooler connection point. \cite{4, 5}

The RRM3 tank contained a sponge propellant management device, controlling approximately 80\% of the tank. When the cryocooler was on, the ullage bubble appeared to be contained above the sponge in the area generally allocated for it. However, when the cryocooler was turned off, the RFMG response changed, more generally agreeing with a model having the ullage near the cryocooler connection. Additionally, temperature sensors indicated a few degrees of heating in the general vicinity of the connection that seemed to be sufficient to surpass the low $\Delta T_{\text{incipience}}$ threshold for methane.

As NASA was beginning to work with DLR to develop the FROST mission \cite{6}, NASA was given ownership of the storage tank. As the storage tank was required to have multiple connections that could provide localized heating, there was a concern for the creation of these hot spots around fluid connections and the cryocooler connection (similar to RRM3). This made it desirable to develop a simple model that could at least bound the cases to understand the local heating and associated stratification that would accelerate the self-pressurization rate of the tank. This was especially important for understanding tank performance during failure modes that might cause loss of power to the spacecraft forcing the cryocooler to be off for extended periods of time.

In addition to understanding the hot spot characteristics, it also became apparent that understanding the system temperature gradients when the cryocooler was active was important. This would help to predict any temperature gradients in the system caused by the thermal restrictions associated with the tank wall making it harder to remove the necessary heat from the storage tank.
2. Model Setup
Two separate computational approaches were taken: (a), a finite element model was developed for wall heat transfer; and (b) three direct relationships provided through analytical solutions of the problem were also developed.

2.1. Finite Element Model
A finite element model, as shown in figure 1, was set up to calculate the temperature gradient due to a local hot or cold spot. The one-dimensional cylindrical coordinate system finite element model has 11 nodes including the central node under the hot/cold spot and the edge node. Heat transfer accounted for in the model includes the energy into or out of the tank wall due to the hot/cold spot, conduction between the cylindrical nodes in the wall, and convection between the wall and the bulk fluid. The heat flux into the fluid (convection) is calculated for each node by solving the conservation of energy for the temperature at each node.

Inputs to the model include the convection heat transfer coefficient (assumed to be constant for all nodes), the diameter of the hot/cold spot, the outer diameter/radius of the calculation (determines the size of each node), the wall thickness and thermal conductivity (assumed to be constant for all nodes), and the heat input/output for the hot/cold spot. Typically, the outer diameter should be sized such that 85 – 95% of the heat is accounted for.

Figure 1. Finite element configuration.

2.2. Analytical Solutions
Several models were formulated that are tractable and amenable to analytical solutions. These include (i) a model equivalent to that solved by the finite element method (ii) one where the heat input is modelled as a point source (iii) an unsteady model with a point heat source in an infinite disk. In all the models, the disk exchanges heat with the bulk fluid with a constant heat transfer coefficient. The heat input at the source is specified. At the outer edge of the disk, the temperature is specified to be equal to that of the bulk fluid. The analytical solutions are summarized here.

2.2.1 Finite heat source
Starting with the one-dimensional equations for radial conduction in a disk with convection on one side, the solution for the temperature at any radial location \( r \) on the disk is determined to be:

\[
T = T_0 + \frac{Q}{2\pi kd} B \left[ K_0 \left( \sqrt{\frac{Bi}{R_0}} \frac{r}{R_0} \right) - \frac{K_0(\sqrt{Bi})}{I_0(\sqrt{Bi})} I_0 \left( \sqrt{\frac{Bi}{R_0}} \frac{r}{R_0} \right) \right], \quad r_0 \leq r \leq R_0
\]

\[
B = \left\{ \frac{r_0}{R_0} \sqrt{\frac{Bi}{I_0(\sqrt{Bi})}} I_1 \left( \sqrt{\frac{Bi}{R_0}} \frac{r_0}{R_0} \right) + K_1 \left( \sqrt{\frac{Bi}{R_0}} \frac{r_0}{R_0} \right) + \frac{1}{2} \left( \frac{r_0}{R_0} \right)^2 Bi \left[ -\frac{K_0(\sqrt{Bi})}{I_0(\sqrt{Bi})} I_0 \left( \sqrt{\frac{Bi}{R_0}} \frac{r_0}{R_0} \right) + K_0 \left( \sqrt{\frac{Bi}{R_0}} \frac{r_0}{R_0} \right) \right] \right\}^{-1}
\]

where \( d \) is the thickness of the disk, \( k \) is its thermal conductivity, \( r_0 \) is the radius of the heat source, \( R_0 \) is the outer radius, \( h \) is the heat transfer coefficient, \( Bi = \frac{hR_0^2}{kd} \) is the Biot number, \( T_0 \) is the bulk fluid temperature, \( Q \) is the heat load, and \( I_n(x) \) and \( K_n(x) \) are Bessel’s functions of order \( n \).
2.2.2 Point heat source
In the limit that \( r_0 \) is zero, the solution given above represents the temperature distribution in the disk subjected to a point source of heat at its center and can be specialized to be:

\[
T = T_0 + \frac{Q}{2\pi kd} \left\{ K_0\left(\sqrt{Bi} \frac{r}{R_0}\right) - \frac{K_0\left(\sqrt{Bi}\right)}{I_0\left(\sqrt{Bi}\right)} I_0\left(\sqrt{Bi} \frac{r}{R_0}\right) \right\}, \quad 0 < r \leq R_0
\]

Note that the value of the temperature at \( r = 0 \) is unbounded; the temperature is physically valid only beyond a small distance away from the center of the disk.

2.2.3 Unsteady solution with a point heat source in an infinite disk
An unsteady solution for a point source of heat in an unbounded disk is also amenable to an analytical solution in the Laplace transform domain and is given here for completeness. Scaled quantities are defined as:

\[
\theta = \frac{T - T_0}{\frac{Q}{2\pi kd}}, \quad r = \frac{r \ast L}{L}, \quad t = \frac{t \ast t_{ref}}{L_{ref}}, \quad L = \left(\frac{kd}{h}\right)^{1/2}, \quad t_{ref} = \frac{\rho C_p d}{h}
\]

The solution for the scaled temperature distribution is

\[
\theta = L^{-1}\left\{ \frac{1}{s} K_0(r\sqrt{1 + s}) \right\}
\]

where \( s \) is the variable in the Laplace domain and \( L^{-1} \) denotes the inverse Laplace transform, that must be obtained by numerical means. The results in the time domain are provided in figure 2.

![Figure 2](image)

**Figure 2.** Time dependent response for transient solution, normalized.

2.3. Comparison of the steady state models
It was important to verify that the finite element and the analytical solutions models produce similar results. This builds some level of confidence that both consider the same physics and are correctly formulated. It does not validate that they provide accurate predictions as both methodologies carry several simplifications including constant material properties, constant fluid heat transfer coefficient, and treating the fluid boundary as an isothermal boundary condition. In reality, there would be some interaction between the temperature gradient in the fluid and the temperature gradient in the wall. Therefore, all three of these assumptions that affect the fluid and wall temperature gradients, limit the accuracy of the results.
Figure 3 shows comparative results between the Finite Element model and the Point Source and Finite Source analytical solutions for inputs of 0.5 W heat load over a 0.1 m diameter source into a 2.54 mm tank wall of conductivity of 55 W/m/K with the fluid sink temperature at 77 K with a convective heat transfer coefficient of 1 W/m²/K. It is seen that both the Point Source analytical solution and the Finite Source analytical solution are in strong agreement with the Finite Element model. These general relationships would be expected based on the assumptions due to the heat input path.

3. Model Results

The models were run for a variety of cases to understand the trends across the different variables and inputs.

3.1. Input Variables

The inputs for the Hot/Cold spot model were varied as shown in Table 1. The heat transfer coefficient ranges in Table 1 were calculated via Rayleigh and Nusselt numbers as shown below the table. The Rayleigh correlation is for a heated/cooled vertical plate. The heat transfer coefficient for a characteristic length of 0.5 m and temperature gradient of 3 K are shown in Fig 4. Scaling to different length or temperature differences should be done by (dT*L)^0.25. For reference, the RRM3 tank wall varied between 3.8 mm and 7.6 mm and was made of a combination of aluminum 6061 and stainless steel 304.

| Variable                                | Hot Spot                          | Cold Spot                      |
|-----------------------------------------|-----------------------------------|--------------------------------|
| Tank Material (thermal conductivity)    | 90 W/m/K (AL 6061)                | 8 W/m/K (SST 304)              |
|                                         | 1 W/m/K (composite)               |                                |
| Heat Transfer Coefficient               | 0.5 – 7.5 W/m²/K                  |                                |
| Heat input/removal rate                 | 0.25 W – 1 W                      | -5 W – (-10)W                  |
| Wall thicknesses                        | 0.00254 m – 0.00635 m (0.1 inch to 0.25 inch) |                                |
| Heat Sink Temperature                   | 77 K                              |                                |
| Cryocooler Connection Diameter          | 0.05 – 0.10 m (2 inch to 4 inch)   |                                |

\[ Ra = GrPr = \frac{g\beta \rho^2 L^3 \Delta T}{\mu^2 Pr} \]
\[ Nu = 0.68 + \frac{0.670 Ra^{0.25}}{1 + \left(0.492 / Pr\right)^{9/16}} ^{4/9} = \frac{hL}{k} \]

Here, \( Ra \) is the Rayleigh Number, \( Gr \) is the Grasshof Number, \( Pr \) is the Prandtl Number, \( Nu \) is the Nusselt number based on correlation from Bergman and Lavine [7] eq. 9.27 for flow over a vertical plate, \( g \) is the local acceleration due to gravity in m/s², \( \beta \) is the coefficient of volume expansion in 1/K, \( \rho \) is the liquid density in kg/m³, \( L \) is the representative length dimension (in this case, tank radius) in meters, \( k \) is the fluid thermal conductivity in W/m/K, \( \Delta T \) is the temperature difference between the tank wall and bulk fluid in K, \( \mu \) is the dynamic viscosity in N·s/m², and \( h \) is the heat transfer coefficient in W/m²/K.

Figure 4. Rayleigh number and heat transfer coefficient for liquid nitrogen at 1 bar as a function of gravitational acceleration.

3.2. Detailed Performance Results
The most important information provided by the models are the predictions of the maximum heat flux and temperature at the local heating spot. These trends are shown for aluminum 6061, stainless steel 304, and a generic composite like material for a 1 W heat input over a 4 cm diameter spot in figure 5 and figure 6. The results reveal that the temperature gradient is significantly more impacted by the material conductivity and heat transfer coefficient than tank wall thickness, though tank wall thickness can still be an important contributor. Additionally, the heat fluxes become quite significant locally at the hot spot for composite tanks while the metallic tanks, especially, the aluminium do a good job of spreading the heat out over a wider area. Figure 7 shows the effect of multiple different heat input rates and also the area of the imposed heat input. Figure 8 shows the temperature penalty that a cooling source encounters while pulling heat of out of a small location as a function of heat transfer coefficient. Specifically, the temperature gradients required to remove the energy are quite large for stainless steel tanks.
Figure 5. Temperature difference as a function of heat transfer coefficient for a 1 W hot spot.

Figure 6. Heat flux as a function of heat transfer coefficient for a 1 W hot spot.

4. Conclusions
It is demonstrated that using an aluminium tank is important, as composite tanks show significant difficulty in distributing the heat flow from point heat loads. The thermal distribution of these spots should be further studied if composite tanks are desired. Additionally, local cooling of large tank at a singular point is highly inefficient, driving large temperature gradients into the system to remove the desired energy. If spot cooling is required for large surfaces where the natural convection heat transfer coefficients are small, care must be taken to design the tank such as not to induce boiling or thermal cooling penalties in these locations.
Figure 7. Heat flux as a function of heat transfer coefficient for a 3.2 mm thick tank wall and different load sizes.

Figure 8. Temperature penalty for a 7.5 W heat removal from aluminum 6061 and stainless steel 304 tank walls.

References

[1] M M Hasan, C S Lin, R H Knoll, M D Bentz, Tank Pressure Control Experiment: Thermal Phenomena in Microgravity, NASA Technical Paper 3564, 1996.

[2] M Kassemi, S Hylton, and O Kartuzova, Results of Microgravity Zero-Boil-Off Tank (ZBOT) Experiment & CFD Model Validation, 26th European Low Gravity Research Association Biennial Symposium and General Assembly and 14th International Conference on “Two-Phase Systems for Space and Ground Applications”, Granada, Spain, September 24-27, 2019.

[3] M Kassemi, O Kartuzova, and S Hylton, Boiling during Self-Pressurization and Pressure Control in the ZBOT Microgravity Experiment, The 28th Space Cryogenics Workshop, Southbury, Connecticut, July 17-19, 2019.

[4] S R Breon et al 2020 IOP Conf. Ser.: Mater. Sci. Eng. 755 012002

[5] G A Zimmerli et al Radio Frequency Mass Gauge (RFMG) Test Results From Robotic Refueling Mission 3 (RRM3) Operations on International Space Station (ISS), NASA TP 2020-5000671, 2020.

[6] T E Bruns et. al., Future-Oriented Research Platform for Orbital Cryogenic Storage Technologies (FROST), presented at the Aerospace Europe Conference 2020, AEC2020-401, 2020.

[7] T L Bergman and A S Lavine, Fundamentals of Heat and Mass Transfer, 8th Edition, Hoboken, NJ: John Wiley & Sons, 2019.

Acknowledgments

This work was funded by the Cryogenic Fluid Management Portfolio Project, a project within the Space Technology Mission Directorate of the National Aeronautics and Space Administration.