Multi-Node Modeling of Cryogenic Tank Pressurization System using Generalized Fluid System Simulation Program

By
Alak Bandyopadhyay
Computer Science, Alabama A & M University, Normal, AL 35762
&
Alok K. Majumdar, Andre C. Leclair and Juan G. Valenzuela
NASA/Marshall Space Center, Huntsville, AL 35812

Extended Abstract

Cryogenic Tanks are pressurized by inert gas such as Helium or Nitrogen to maintain the required pressure of the propellant delivered to the turbo-pump of a liquid rocket engine. Thermo-fluid system simulation tools are used to analyze the pressurization process of a cryogenic tank. Most system level codes (GFSSP and ROCETS) use single node[1] to represent ullage which is the gaseous space in the tank. Ullage space in a cryogenic tank is highly stratified because the entering inert gas is at ambient temperature whereas the liquid propellant is at a cryogenic temperature. A single node model does not account for the effect of temperature gradient in the ullage. High fidelity Navier-Stokes based CFD model of Tank Pressurization is not practical for running a long duration transient model with thousands and millions of nodes. A possible recourse is to construct a multi-node model with system level code that can account for ullage stratification.

For the past few years, United Launch Alliance has been developing a propulsion system called Integrated Vehicle Fluids (IVF) to improve the functional and reliability limits of upper stages for long-duration space missions. IVF uses boil-off propellants to drive thrusters for the reaction control system as well as to run small internal combustion engines (ICEs). The produced thrust is used for maneuvering the vehicle and to settle propellants during coast flight. Figure 1 shows a simplified schematic of the IVF system including the propellant tank and a fluid loop consisting a compressor and heat exchanger instead of a helium tank in a conventional propulsion system. The compressor intakes propellant vapor from the tank ullage and drives it through a heat exchanger to heat it before it sends it back to the tank for pressurization. The heat exchanger receives heat from coolant of the ICE. The ICE provides power to the compressor and battery. The network flow solver program GFSSP[2] has been used to model the heat exchanger component and the complete IVF system by using one dimensional model (changing only in the tank axial direction) for temperature and pressure by LeClair et.al.[3] and Majumdar et.al.[4]. However both these models are unable to see any two dimensional effect within the tank.

The objective of the current study is to develop a multi-node computational model to simulate the pressurization of the tank due to the propellant injection from the top of the tank and vent off after some time. In the current model, the IVF loop is excluded. The testing data are available with liquid Nitrogen; hence, in the current model liquid N_2 has been used as the working fluid. The model also considers the conjugate heat transfer in the tank wall.

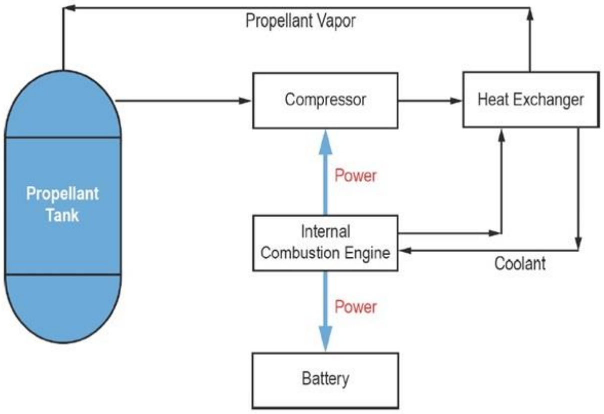

Figure 1. Simplified Schematic of IVF System

The tank is 10 ft. tall and core diameter is close to 10 ft. (119 inches) and the dome diameter at the top is 77 inches. The tank is assumed 75% filled with liquid N_2. The tank wall consists of stainless steel on the inside and surrounded by foam insulation on the outside. The ambient node is assumed to be at 86 F.

Figure 2 below shows the computational model using the geometry set up feature of the network flow solver (GFSSP), by using 25 fluid nodes (5 nodes in the horizontal or radial direction, and five in vertical sections) in the tank and 20 solid nodes for the tank wall. Nodes 21, 22, 23, 24 and 25 are the nodes close to top of the tank (in the ullage space) and nodes 1, 2, 3, 4, 5 are fluid nodes close to the liquid surface. The left side of the figure 2 is close to center line of the tank. The model is assumed to be axisymmetric with varying tank radius in the axial (vertical) direction.

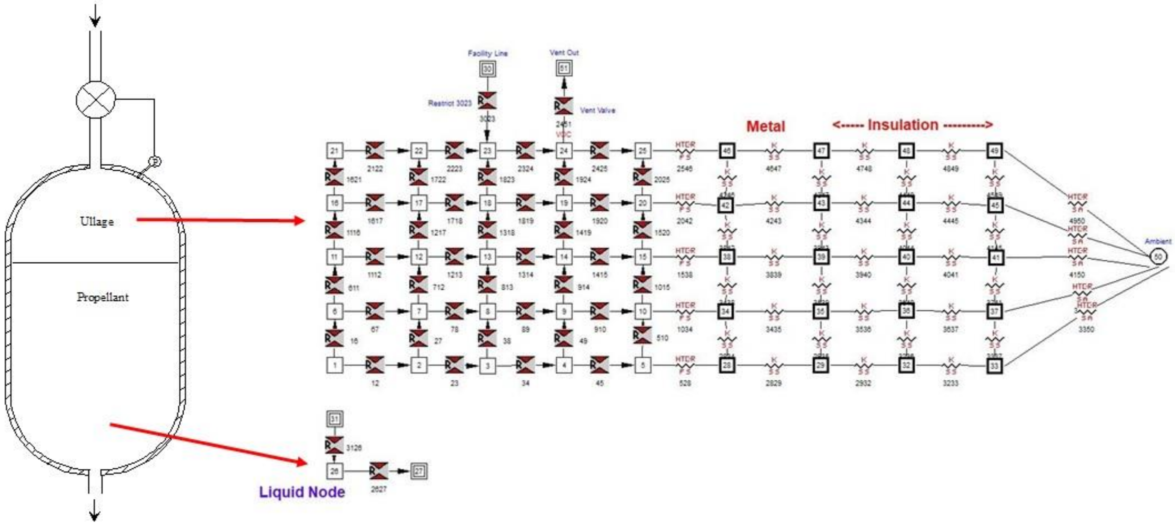

Figure 2. Multi-Node Model of the Propellant Tank

In the current model, the governing differential equations (mass, momentum, energy) are solved in the ullage space with mass transfer between the liquid (represented by node 26) and the ullage through the user subroutine. The liquid-vapor interaction modeling for heat and mass transfer is described in detail by Majumdar et. Al[4]. Node 30 represents the propellant injection point at the top of the tank and node 51 represent the outside boundary representing the venting out of the propellant. The heat transfer coefficient between the wall and ullage was computed from a natural convection correlation for a vertical plate[5]. The pressure and temperature at which the propellant is injected at the tank top and the vent valve open history are shown as a function of time in Figures 3(a) and 3(b) respectively. The liquid temperature is at -315.2 F.

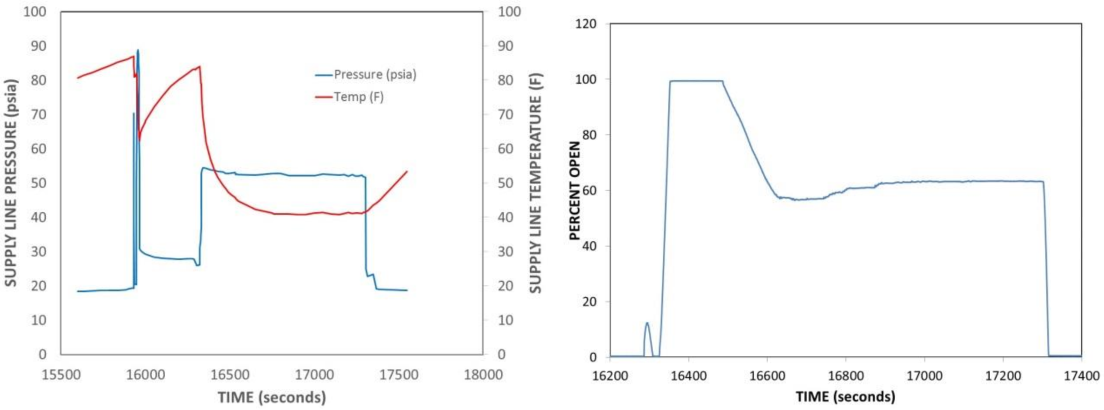

Figure 3. (a) Pressure and Temperature of the injected propellant and (b) vent valve opening

Both the simulations and test data confirm that the ullage pressure does not vary appreciably along either radial or axial direction and only varies with time. However the temperature is stratified along the tank height, and at the same time it varies in the radial direction. Figure 4(a) and 4(b) show the comparison of computed data with the test data for pressure and temperature respectively. The predictions match quite well with the test data.

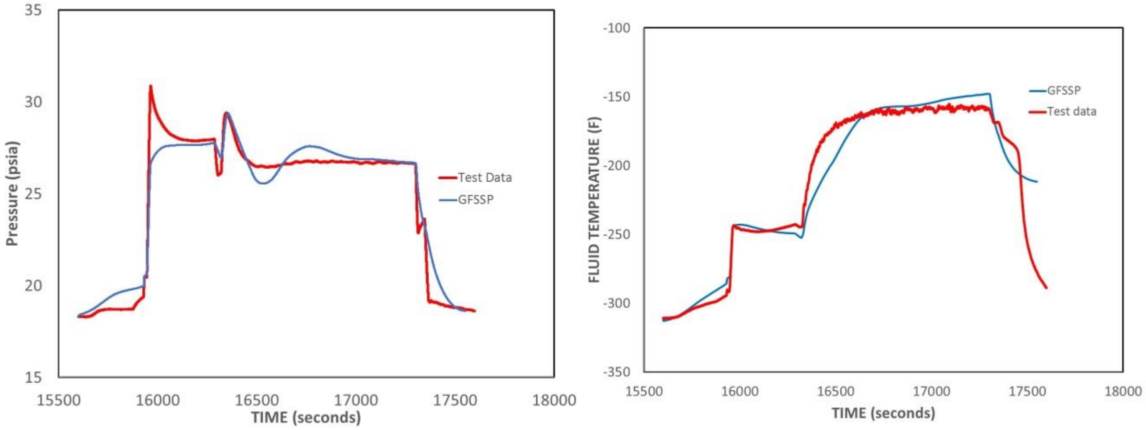

Figure 4. Comparisons of Computed with Test Data for (a) Ullage Pressure and (b) Ullage Temperature at a reference node (node 18).

Figure 5 shows the temperature contours (colored domain) and the streamline plots in the ullage space illustrating the stratification as well as circulation largely due to natural convection.

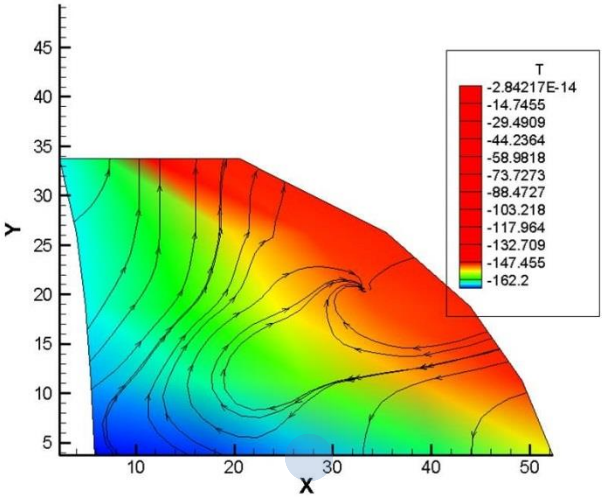

Figure 5. Predicted Temperature Contour and Stream Traces in the Ullage

The extended abstract describes the model results and comparison with test data at 75% fill level. The full paper will also include the model results and its verification with test data in other fill level.

References

[1] Majumdar, Alok and Steadman, Todd, ”Numerical Modeling of Pressurization of a Propellant Tank”, Journal of Propulsion and Power, Vol 17, No.2. March – April, 2001.

[2] Majumdar, A. K., LeClair, A. C., Moore, R., and Schallhorn, P. A., “Generalized Fluid System Simulation Program, Version 6.0,” , *NASA/TP—2016–218218,* March 2016.

[3] Majumdar, A.K, LeClair, A.C. and Hedayat, A., “Numerical Modeling of Pressurization of Cryogenic Propellant Tank for Integrated Vehicle Fluid System”, AIAA Propulsion and Engineering Forum, 52nd *AIAA/SAE/ASEE Joint Propulsion Conference*, July 25-27, 2016, Salt Lake City, UT, pp 1-18

[4] LeClair, A.C., Hedayat, A. and Majumdar A.K., “Numerical Modeling of an Integrated Vehicle Fluids System Loop for Pressurizing a Cryogenic Tank”, AIAA Propulsion and Engineering Forum, 53rd *AIAA/SAE/ASEE Joint Propulsion Conference,* 10-12 July 2017, Atlanta, GA

[5] Rohsenow WM, Hartnett JP, Cho YI., *Handbook of heat transfer*. 3rd edition,, McGraw-Hill; 1998. p. 4.13.