Active Thermal Paper Session Abstracts

Active Thermal Paper Session #1 (Tuesday 8:00 AM to 12:00 PM)

ID	Title	Author(s)	Affiliation	Email
TFAWS2011-AT-001	An Active Broad Area Cooling Model of a Cryogenic Propellant Tank with a Single Stage Reverse Turbo-Brayton Cycle Cryocooler	M.C.Guzik, T.M.Tomsik	GRC	monica.c.guzik@nasa.gov
TFAWS2011-AT-002	Modeling a Cryogenic Tank Pressurization with Active Thermal Cooling Transient Profile Using Excel and Visual Basic for Applications	M.C.Guzik, J.P.Elchert	GRC	monica.c.guzik@nasa.gov
TFAWS2011-AT-003	Broad Area Cooler Concepts for Cryogenic Propellant Tanks	R.J. Christie, T.M. Tomsik, J.P.Elchert, M.C.Guzik	GRC	robert.j.christie@nasa.gov
TFAWS2011-AT-004	Behavior of the GOES-R LHP and Thermal Bus in Vacuum Testing	Perry G Ramsey, Kirsten S Swanson, Behrooz Ghaffarian, Shikha Ganguly	ITT Geospatia l Systems	perry.ramsey@itt.com
TFAWS2011-AT-005	Generalized Fluid System Simulation Program (GFSSP) Version 6 - General Purpose Thermo-Fluid Network Analysis Software	Alok Majumdar, Andre Leclair, Ric Moore, Paul Schallhorn	MSFC	alok.k.majumdar@nasa.gov
TFAWS2011-AT-006	Comparsion of Surface Area and Pumping Power Requirements in Heat Exchangers using Nanofluids and Basefluids Suitable for NASA Applications	Ravikanth S. Vajjha, Debendra Das	University of Alaska	dkdas@alaska.edu

Active Thermal Paper Session #2 (Friday 8:00 AM to 12:00 PM)

ID	Title	Author(s)	Affiliation	Email
TFAWS2011-AT-007	Online Approximation Assisted Optimization and CFD Verification of Microchannel Designs	Khaled Saleh, Vikrant Aute, Reinhard Radermacher, Shapour Azarm	University of Maryland	vikrant@umd.edu
TFAWS2011-AT-009	Hypersonic Vehicle Thermal Management System Assessment	Michael Seal, Tom Lavelle	LaRC	michael.d.seal@nasa.gov
TFAWS2011-AT-010	Venting of a Water/Inhibited Propylene Glycol Mixture in a Vacuum Environment – Characterization and Representative Test Results	Eugene K. Ungar, Lisa R. Erickson	JSC	eugene.k.ungar@nasa.gov
TFAWS2011-AT-011	Role of Direct Numerical Simulation of Multiphase Flow in Fluid	Sadegh Dabiri	University of Notre Dame	sdabiri@nd.edu

TFAWS-AT-001

AN ACTIVE BROAD AREA COOLING MODEL OF A CRYOGENIC PROPELLANT TANK WITH A SINGLE STAGE REVERSE TURBO-BRAYTON CYCLE CRYOCOOLER

M. C. Guzik, NASA Glenn Research Center
T. M. Tomsik, NASA Glenn Research Center

ABSTRACT

As focus shifts towards long-duration space exploration missions, an increased interest in active thermal control of cryogenic propellants to achieve zero boil-off of cryogens has emerged. An active thermal control concept of considerable merit is the broad area cooling of a cryogenic propellant tank with an integrated cooling system that can be used to achieve reduced and zero boil-off of in-space cryogens. One such cooling system is the reverse turbo-Brayton cycle cryocooler. This system is unique in that it has the ability to efficiently provide both cooling and circulation of the coolant gas in the same loop as the broad area cooling lines, allowing for a single cooling gas loop, with the primary heat rejection occurring via a radiator/aftercooler. Currently few modeling tools exist that can size, scale, and characterize an integrated reverse turbo-Brayton cycle cryocooler with broad area cooling design. This paper addresses efforts to create such a tool using Microsoft Excel with Visual Basic for Applications (VBA) in order to gain a broader understanding of these systems, and investigate their performance in potential space missions. The model uses conventional engineering and thermodynamic relationships to predict the preliminary design parameters such as input power requirements, pressure drops, cycle performance, cooling lift, broad area cooler line sizing, and component operating temperatures and pressures given the cooling load operating temperature, heat rejection temperature, compressor inlet pressure, compressor rotational speed, and cryogenic tank geometry. At the time this paper was written, the model was verified to match existing theoretical documentation within a reasonable margin. While further experimental data is needed for full validation, this tool has already made significant steps towards giving a clearer understanding of the performance of a reverse turbo-Brayton cycle cryocooler integrated with broad area cooling technology for zero boil-off active thermal control.

Back

TFAWS-AT-002

MODELING A CRYOGENIC TANK PRESSURIZATION WITH ACTIVE THERMAL COOLING TRANSIENT PROFILE USING EXCEL AND VISUAL BASIC FOR APPLICATIONS (VBA)

M. C. Guzik, NASA Glenn Research Center
J. P. Elchert, NASA Glenn Research Center

ABSTRACT

As interest develops in the area of zero boil-off cryogenic propellant storage, the need to visualize and quantify cryogen behavior during ventless tank self-pressurization and subsequent cool-down with active thermal control has become apparent. During the course of a mission, there is a high probability that a situation will occur that requires active cooling to be stopped for some time. Numerous efforts have been made to characterize cryogenic tank pressurization during ventless cryogen storage without active cooling, but few tools exist to model this behavior in a user-friendly environment for general use, and none exist that quantify the marginal active cooling system size needed for power down periods to manage tank pressure response once active cooling is resumed. By combining the power and flexibility of Visual Basic for Applications (VBA) with the Microsoft Excel platform, a universal, in-depth model can be placed in a simple, easily navigable format. This paper describes a Transient pressurization with Active Cooling Tool (TACT) based on a three-lump homogeneous thermodynamic self-pressurization model, created in Excel using the VBA developer. TACT's capabilities include parametric studies of various cooling lifts of the active cooling system, comparisons of two different fluids and tank materials, and variable heating profiles during the power-off period. By receiving input data on the tank material and geometry, propellant initial conditions, and both the passive heat leak and transient heating rate during self-pressurization, it displays a pressurization profile, including the maximum tank internal pressure and the total time needed to return to the designated start-box pressure. The tool's flexible VBA coding provides the ability to expand in scope to accommodate increasingly detailed mission profiles while remaining an effective but simple tool to aid in understanding and characterizing transient cryogenic pressurization profiles for engineering applications.

Back

TFAWS-AT-003

BROAD AREA COOLER CONCEPTS FOR CRYOGENIC PROPELLANT TANKS

R. J. Christie, NASA Glenn Research Center
J. P. Elchert, NASA Glenn Research Center
M. C. Guzik, NASA Glenn Research Center

ABSTRACT

Several studies have shown that Broad Area Cooling (BAC) is an effective way to intercept heat before it enters the fluid in an in-space cryogenic propellant storage tank. If enough heat can be intercepted, boil-off of cryogenic propellants can be eliminated or at least greatly reduced. This would enable long duration storage of cryogenic propellants in-space. A Broad Area Cooling system typically consist of a cryocooler, a circulator and a network of cooling tubes and possibly a cooled shield. Such a system might also include interface heat exchangers and recuperators. Although there are different types of cryocoolers and circulators, some cryocoolers can provide both the refrigeration and the circulation functions. This study looks at the numerous hardware combinations that can be used to build a Broad Area Cooler, which 'flight like' components are currently available, and what is their Technical Readiness Level (TRL). This study also considers single phase and two-phase systems and various fluid types. The study concludes that several combinations of cryocoolers and circulators can be used to build a BAC. Therefore a preferred system would be one with a high TRL and one that can be scaled to provide cooling capacities on the order of 150W at 90K.

Back

TFAWS-AT-004

BEHAVIOR OF THE GOES-R LHP AND THERMAL BUS IN VACUUM TESTING

Perry G. Ramsey, ITT Geospatial Systems
Kirsten S. Swanson, ITT Geospatial Systems
Behrooz Ghaffarian, ITT Geospatial Systems
Shikha Ganguly, ITT Geospatial Systems

ABSTRACT

Thermal vacuum testing of the Advanced Baseline Imager (ABI) Prototype Module (PTM) was completed on November 6, 2010. The ABI thermal system uses a centralized thermal bus concept, incorporating a network of constant conductance heat pipes to collect heat from sources throughout the instrument sensor unit. The bus is cooled by a fully redundant temperature controlled LHP/radiator.

The performance of the LHP radiator system has previously been reported; therefore, this paper focuses on unique behaviors of the integrated system. Overall, the thermal system performed very well, managing instrument temperatures as commanded.

One interesting behavior of a non-operating loop is to start at a lower temperature than the control temperature of the compensation chamber. This is shown to be related to the details of the thermal bus producing non-uniform temperatures within the evaporator, thus leading to easier startup than would ordinarily be expected.

The behavior of the loop under control of an on/off heater is complex. Although the evaporator temperature remains within narrow limits, large variations in the compensation chamber temperature are observed, variations that are significantly larger than the dead band of the heater. Evidence will also be presented indicating backwards flow under certain heater operations.

Back

TFAWS-AT-005

GENERALIZED FLUID SYSTEM SIMULATION PROGRAM (GFSSP) VERSION 6 – GENERAL PURPOSE THERMO-FLUID NETWORK ANALYSIS SOFTWARE

Alok Majumdar, NASA Marshall Space Flight Center
Andre Leclair, NASA Marshall Space Flight Center
Ric Moore, NASA Marshall Space Flight Center
Paul Schallhorn, NASA Kennedy Space Center

ABSTRACT

Generalized Fluid System Simulation Program (GFSSP) is a general purpose thermo-fluid network analysis software developed at NASA/Marshall Space Flight Center and Kennedy Space Center for Main Propulsion System, Turbo-machinery and Launch Service Program analysis. This paper will describe the additional capabilities of Version 6 which has been recently released for Beta Testing. The additional capabilities include: a) Fluid mixture option that allows phase change in one of the constituents of the mixture; b) Pressure regulator option with forward looking algorithm,; c) Prescribed flow option in conjunction with prescribed pressure option; d) Two dimensional Navier Stokes solver in Cartesian and cylindrical polar co-ordinate in a component of a fluid system network. The solution algorithm, application and verification of additional capabilities will be described. The paper will also address the future development activities planned for the code.

Back

TFAWS-AT-006

COMPARISON OF SURFACE AREA AND PUMPING POWER REQUIREMENTS IN HEAT EXCHANGERS USING NANOFLUIDS AND BASEFLUIDS SUITABLE FOR NASA APPLICATIONS

Ravikanth S. Vajjha, University of Alaska Fairbanks
Debendra K. Das, University of Alaska Fairbanks

ABSTRACT

Nanofluids are dispersions of nanometer size particles, normally less than 100nm, in conventional heat transfer fluids. Substantial research in recent years has shown that nanofluids represent a superior heat transfer medium, in comparison to the conventional basefluids in use today. Therefore, the adoption of nanofluids has the potential benefit of reducing the size and weight of heat transfer systems, while requiring less pumping power to transfer the same amount of heat. This effect will have noticeable benefits on NASA missions, such as the Orion vehicle and the space lab, where the heat transfer needs are abundant. Being motivated by this benefit, we have computed the hydraulic and thermal performances of three different nanofluids containing aluminum oxide, copper oxide and silicon dioxide nanoparticles in a basefluid. From the analysis, it is observed that, given a constant amount of heat transfer in a liquid to air heat exchanger, the surface area and the pumping power necessary by the three nanofluids are lesser than that required by the basefluid. Tabular results will be presented to compare the performance of three nanofluids with the basefluid and also with one another. The thermophysical properties equations, such as those of viscosity, thermal conductivity, specific heat and density for the nanofluids will also be summarized. Copper oxide nanofluid provides the highest convective heat transfer compared to the other two and the basefluid. However, it also requires the highest pumping power due to its higher viscosity. And viscosity is strongly dependent upon the volumetric concentration of nanoparticles in the nanofluid. All these aspects will be presented and the strength and weakness of nanofluids versus the basefluid will be discussed.

Back

TFAWS-AT-007

ONLINE APPROXIMATION ASSISTED OPTIMIZATION AND CFD VERIFICATION OF MICROCHANNEL DESIGNS

Khaled Saleh, University of Maryland
Vikrant Aute, University of Maryland
Reinhard Radermacher, University of Maryland
Shapour Azarm, University of Maryland

ABSTRACT

Microchannel based heat sinks have been shown to generate significant heat transfer rates from extremely small volumes and hence are used in many electronic cooling devices in high heat flux applications. These applications include, but are not limited to, power electronics, high heat load optical components, and hybrid vehicle power electronics. The selection of an appropriate microchannel heat sink and the determination of its optimal design can lead to significant improvements in the overall performance and heat density.

This paper considers a microchannel design optimization problem with the objective of minimizing the maximum channel temperature while maximizing the heat rejected per unit volume of the microchannel. This is a two-objective optimization problem resulting in a tradeoff between the aforementioned two conflicting objectives. In order to find optimally compact heat exchanger designs, mini and microchannel geometries are being considered. The goal in this paper is to evaluate the potential in designing high heat density microchannels for a given application and at the same time reduce the computational effort required to do so. A newly developed online approximation assisted optimization technique is applied to optimally design a microchannel with single phase flow and constant heat flux. Adaptive sampling technique and a Kriging metamodeling method are used to build metamodels for the maximum temperature, fluid outlet temperature, and pressure drop inside the channels based on CFD analysis. A multi-objective genetic algorithm (MOGA) is used as the optimizer. An online approximation technique is used combined with MOGA to update the metamodels with new samples with the purpose of improving the metamodel performance in the expected optimum regions as well as finding better designs. The optimum solutions are verified using CFD simulations. It is observed that online approximation assisted optimization obtains reasonably accurate optimum design solutions while reducing significantly the computational time.

Back

TFAWS-AT-009

HYPERSONIC VEHICLE THERMAL MANAGEMENT SYSTEM ASSESSMENT

Michael Seal, Analytical Mechanics Associates
Tom Lavelle, NASA Glenn Research Center

ABSTRACT

This document describes the development and assessment of a hypersonic vehicle thermal management system. The scramjet, high-speed flowpath planned for the NASA reference TBCC-based TSTO vehicle concept requires the use of actively-cooled surfaces for the majority of the flowpath wetted area. Information presented provides a description of the thermal management system (TMS) for this high-speed flowpath and associated propellant feed systems, including system definition and layout, thermal and power balance analyses, structural concept assessment and system/component sizing.

Back

TFAWS-AT-010

VENTING OF A WATER/INHIBITED PROPYLENE GLYCOL MIXTURE IN A VACUUM ENVIRONMENT – CHARACTERIZATION AND REPRESENTATIVE TEST RESULTS

Eugene K. Ungar, NASA Johnson Research Center
Lisa R. Erickson, NASA Johnson Research Center

ABSTRACT

A planned use of the Orion space vehicle involves its residence at the International Space Station for six months at a time. One concept of operations involves temporarily venting portions of the idle Orion active thermal control system (ATCS) during the docked phase, preventing freezing. The venting would have to be reasonably complete with few, if any, completely filled pockets of frozen liquid. Even if pockets of frozen liquid did not damage the hardware during the freezing process, they could prevent the system from filling completely prior to its reactivation.

The venting of single component systems in a space environment has been performed many times and is well understood. Because the Orion ATCS working fluid is a 50/50 mixture of water and inhibited propylene glycol, its boiling behavior was expected to differ from that of a pure fluid. A test was developed to compare the evaporation behavior of pure water, a 50/50 mixture of water and inhibited propylene glycol, and inhibited propylene glycol.

The test was performed using room temperature fluids in an insulated thin walled stainless steel vertical tube. Reticulated polyurethane foam was placed inside the tube to reduce the convection currents. A vacuum system connected to the top of the tube set the pressure boundary condition. The mass of the test article was measured as it changed over time, as was its temperature and backpressure.

The tests were successful. Somewhat surprisingly, the results showed that the evaporation behavior of the three fluids had more similarities than differences. The 50/50 mixture evaporated similarly to the pure water – albeit at a slower rate. The test results indicate that our extensive space-based experience with venting of single component fluids can be applied to the problem of Orion ATCS venting.

Back

TFAWS-AT-011

ROLE OF DIRECT NUMERICAL SIMULATION OF MULTIPHASE FLOW IN FLUID/THERMAL ANALYSIS

Sadegh Dabiri, University of Notre Dame
Jiacai Lu, University of Notre Dame
Gretar Tryggvason, University of Notre Dame

ABSTRACT

During the last decade, DNS of multiphase flows has emerged as one of the most promising approaches to understand multiphase flow physics. Direct numerical simulation of the Navier-Stokes equations provides essentially exact solutions to multiphase flow problems. These solutions can be used to understand the fundamental aspects of multiphase flows and to provide a database for developing and validating models of small-scale unresolved motion in simulations of industrial processes. Here, we discuss DNS results for two different multiphase flow problems, turbulent bubbly flow in vertical channels, and nucleate and film boiling. In the former, the effects of bubble deformability and flow direction on the dynamics of the flow will be explained. For boiling we show results for the formation of several vapor bubbles and discuss the challenges of resolving the small length and time scales present during nucleation.
Back