Upper Stage Tank Thermodynamic Modeling Using SINDA/FLUINT

Paul Schallhorn*, D. Michael Campbell**, Sukhdeep Chase**, Jorge Piquero**, Cindy Fortenberry**, Xiaoyi Li**, and Lisa Grob***

* NASA Kennedy Space Center
Kennedy Space Center, FL

**Analex Corporation
Kennedy Space Center, FL

***Edge Space Systems
Glenelg, MD

ABSTRACT

Modeling to predict the condition of cryogenic propellants in an upper stage of a launch vehicle is necessary for mission planning and successful execution. Traditionally, this effort was performed using custom, in-house proprietary codes, limiting accessibility and application. Phenomena responsible for influencing the thermodynamic state of the propellant have been characterized as distinct events whose sequence defines a mission. These events include thermal stratification, passive thermal control roll (rotation), slosh, and engine firing. This paper demonstrates the use of an off the shelf, commercially available, thermal/fluid-network code to predict the thermodynamic state of propellant during the coast phase between engine firings, i.e. the first three of the above identified events. Results of this effort will also be presented.

Introduction

Knowledge of the thermodynamic state of cryogenic propellant is necessary for a successful engine start and operation. In the case of an upper stage, the requisite source of this knowledge is a series of analytical predictions involving thermal radiation, conduction, convection, evaporation, recirculation and mixing. In many cases, this involves a series of separate modeling tools to account for these items, i.e. a thermal structural modeling tool, a thermodynamic tool, a computational fluid dynamics tool, etc. A downfall of this approach is the information exchange between these separate modeling tools (and their respective modelers). Due to various constraints, the information loop between these separate modeling tools is not iterated until convergence. The result is that the prediction of the thermodynamic condition of the propellant is likely inaccurate.

A new approach, attempting to combine several of the separate modeling tools, using a thermal/fluid network code is proposed. The primary purpose is to allow the closely coupled thermal and fluids/thermodynamics aspects to be solved simultaneously, resulting in a higher fidelity modeling prediction of the thermodynamic state of the propellant. After review of the available thermal and fluid network codes, a decision to use SINDA/FLUINT[1] (an industry standard, commercially available package) was made.

SINDA/FLUINT is comprised of two distinct modeling tools which share and exchange information during each iteration and/or time-step. The SINDA portion was originally developed in the 1960's and has undergone several major modifications during the 1970's and 1980's, as well as continuous improvements. Its primary purpose is to solve the heat equation featuring "nodes" and "conductors." The FLUINT portion was added in the later half of the 1980's and, like its SINDA counterpart, has undergone continuous improvement. FLUINT solves the one-dimensional continuity, momentum and energy equations for an internal flow condition using "lumps" and "paths." Energy is exchanged between SINDA and FLUINT using a component in FLUINT called a TIE. Each TIE provides a convective heat transfer connection between one fluid lump to one thermal node (note that both lumps and nodes can have multiple TIEs - i.e. one lump may have TIEs to many nodes and vice-versa).

The open literature contains limited information[2-4] on low gravity cryogenic propellant conditioning (thermal stratification, effects of rotation, etc.). In an attempt to anchor the proposed modeling approach, a parallel effort was undertaken by the Florida Institute of Technology[5,6] to incorporate various physical phenomena (low gravity, rotation, stratification, etc.) into the analytical prediction of the propellant's thermodynamic state.

Approach

The modeling for this effort is broken down into distinct categories separated by medium (solid/fluid).

The solid (structural) model is straight forward, following traditional modeling methodologies. Thermal conduction and radiation models are developed for the upper stage in question. Development is performed using a CAD based GUI (Thermal Desktop[6]) with corresponding radiation model analysis tool (RadCAD). Since the fluid portion of the modeling is performed outside the normal bounds of a flow network code, a decision was made early in the planning phase of the modeling effort to forgo the fluid network capability within Thermal Desktop (FloCAD) and instead output the SINDA model and manually include the thermodynamic modeling. As part of this development, the thermal structural and thermodynamics modelers engaged to decide upon the number of axial layers and the number of circumferential divisions needed within each tank model. This allows the thermal structural model and the thermodynamic model to be integrated easily.

The fluid (thermodynamic) modeling is broken down into distinct "events," which include settled thermal stratification, rotation, and slosh. Figure 1, below, illustrates these events. The resolution of each event differs from the others, with the stratification event having the most resolution (in order to predict the thickness and thermal variation within the stratum) and the slosh event having the least (slosh is considered to be a mixing event, therefore, tracking of warm and cold propellant is of lesser importance). A fourth event model (pull through) is slated to be developed and incorporated into the stratification event model at a later date.

CFD was used as input to the thermodynamics model with location of the fluid within the tanks as a function of mission time and event (i.e., sloshing).

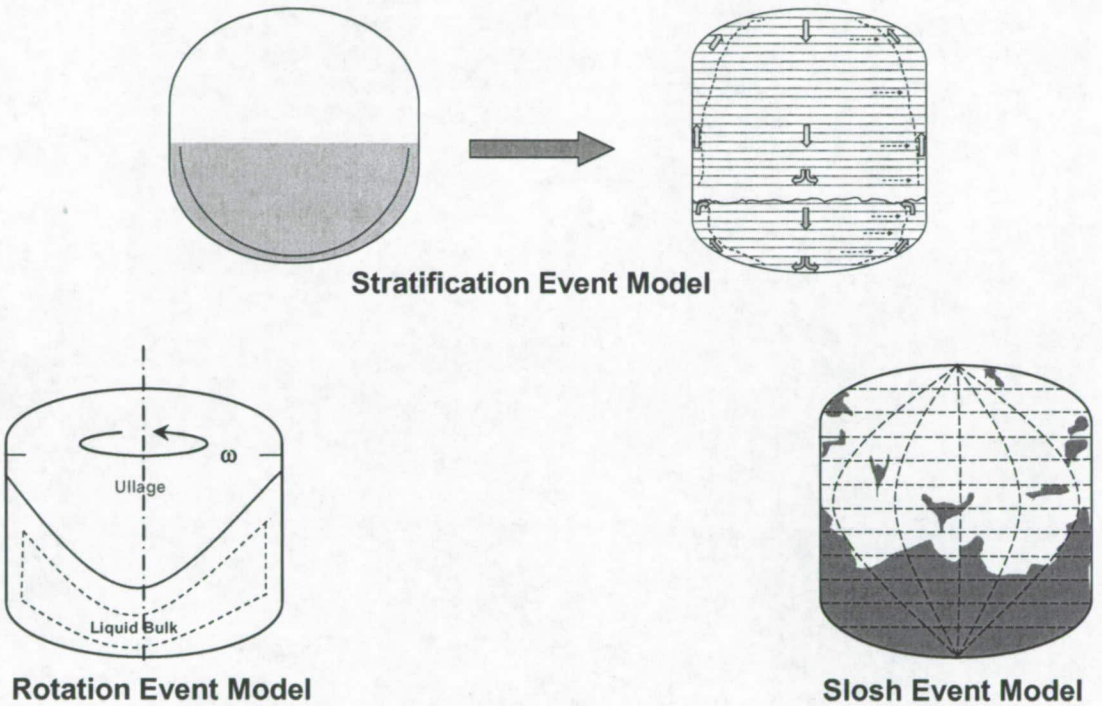

Figure 1: Fluid "Event" Models

Prior to incorporation into the integrated thermal/thermodynamic models, each of the event models were developed for a "simple, generic" tank in order to easily develop and test the various aspects of each event. These aspects account for the physics to be modeled and include: stratification, rotation (both solid body and transients – startup, roll reversal & despin), boiling, molecular diffusion, pressurization and venting, etc.

A note on what follows: from this point forward, the paper was prepared with an assumption that the reader has some familiarity with SINDA/FLUINT and its network building components (nodes, conductors, lumps, connections, etc.). Thus, no detailed

explanation is provided for modeling parameter choices of the commercial code used in this effort.

Methodology

A few points are worth mentioning on the overall method chosen to "integrate" the models. Thermal structural models are present continuously throughout a mission simulation. These models define the environments surrounding the two cryogenic propellant tanks through the evaluation of heat leaks (structural conduction paths) and the heat flux to the general insulation external to the tank wall. The continuous presence of the environmental models is in contrast to the manner in which the fluid models are used.

Modeling of the fluid thermodynamics is accomplished through the use of fluid (FLUINT) sub-models intended to simulate discrete phenomena descriptive of fluid behavior in a space vehicle during multiple engine firings and coast periods. Modeled fluid behaviors are referred to as "events" due to their association with discrete periods of mission time and vehicle behavior. The fluid events which have been identified at this time are a) rotation, b) stratification and c) sloshing (splashing).

Fluid Submodel Integration (Spatial, Temporal)

A) Fluid to structure integration: Fluid models require to be connected (TIE) to the thermal structural heat transfer models and accounted for in appropriate SINDA logic, data and output blocks (i.e. LO2 and LH2 stratification and rotation models are connected at the wall to their respective heat transfer models).

B) Transient integration: The processes of engaging and disengaging individual fluid sub-models to simulate discrete "events" along a continuous time line is accomplished with successive 'Build' commands. Models previously built, but not included in a current build command, are placed into a 'dormant state'. Sequencing of events is controlled in the "OPERATIONS" block. The method used depends upon a) knowledge of the mission being simulated, b) interpretation/identification of environments which signify an event, c) the use of multiple definitions of simulation completion times enables the transition between events and d) identification of variables necessary to maintain continuity between events (passing needed values from the discontinued event to the current event).

There are possible alternatives to the method described in item "c" above. Basically, the current method relies upon an interpretation of the environments (ambient environment, vehicle behavior, etc.) before hand in order to define the required "TIMEND" for each event. An alternative method would be to key the use of "event" sub-models to values of specific variables (example: rotation rate, heat transfer regimes, etc. This would require a real time evaluation of particular mission variables. An examination of real space vehicle mission data quickly dismissed this idea. For example, a rotation speed of zero degrees per second may indicate a slosh event or an opportunity for fluid stratification. Use of the correct event model may require a judgment based upon the foreknowledge of the length of time the event occurs.

Miscellaneous

In addition to use of individual sub-models, much of the required input data is modularized. The program utilizes multiple inserts of external data files used to define a) mission variables (gravity, rate of rotation, vent schedule, etc.), b) tank geometry, c) provide temporary storage, d) fluid properties used in subroutines, and e) CFD data relevant to fluid behavior (occurrence of slosh and location of fluid due to slosh, rotation and gravity forces).

The emphasis herein is to review the method of modeling these key events which together may be used to account for a variety of possibly mission scenarios. However, subroutines created for this modeling effort provide for a number of functions common to all the event sub-models. Some of these functions may be routine (determination of wall node positions, overall liquid depth, etc.) while others provide for the determination of boundary layer and convective heat transfer variables. Because of their importance, a summary of the equations used for the determination of relevant natural convection, boiling and boundary layer variables is provided in Appendix A.

Molecular diffusion (Appendix A), venting and pressurization are included in the modeling of the tank thermodynamics. Molecular diffusion is accounted for in determining the condition of the ullage in conjunction with the use of helium as a pressurant. Molecular diffusion is not a separate 'event' and is not a separate subroutine. Venting and pressurization of the tanks are modeling using simple control valve connectors. Controlling the venting and pressurization to correspond to proper values of valve seat and crack pressures is performed within the flow logic block inherent in SINDA/FLUINT. The models are capable of using multiple values of seat and crack pressure control settings. The varying of control valve settings, which may occur during a single mission for the purpose of propellant conditioning, is controlled with the use of a 'que' defined in an array versus mission time.

Stratification Event

Thermal stratification is probably one of the subjects most likely to surface when a discussion of cryogenic tanks comes to mind. For this reason, stratification was chosen as the first 'event' to be modeled. The typical 'top to bottom' existence of temperature stratum within a fluid is largely due to buoyancy driven forces which relocate fluid within the boundary layer to upper stratum and eventually to the liquid-vapor interface. The migrating boundary layer fluid creates a 'warm layer' at the liquid vapor interface. The ability to predict the existence and the extent of temperature gradients within the fluid is essential to understanding phenomena such as tank pressure control, boil off and the possible variance in fuel temperatures at engine and pump inlets. It is important to note that the stratification model is built so that heat transferred to the fluid at the wall may be simulated using either an assumed direct heat input, an assumed temperature difference ($T_{WALL} - T_{FLUID}$) or through a 'TIE' to nodes of a thermal heat transfer model of the surrounding structure.

Conceptually, the model needs to 1) account for the energy and mass transport mechanisms responsible for thermal stratification and 2) exhibit fidelity fine enough to capture thermal stratification. Key locations where mass and energy transport require attention are at the vessel wall and at the liquid vapor interface (Figure 2). Various regimes of convective heat transfer between the vessel wall and the fluid are accounted for through the use of separate user written subroutines for natural convection and boiling. These subroutines, located in the subroutine data block, are common to all the event models. Despite SINDA/FLUINT having in house subroutines available, these subroutines were written to afford more control over the correlations used, variables desired, and the accounting for micro-gravity effects.

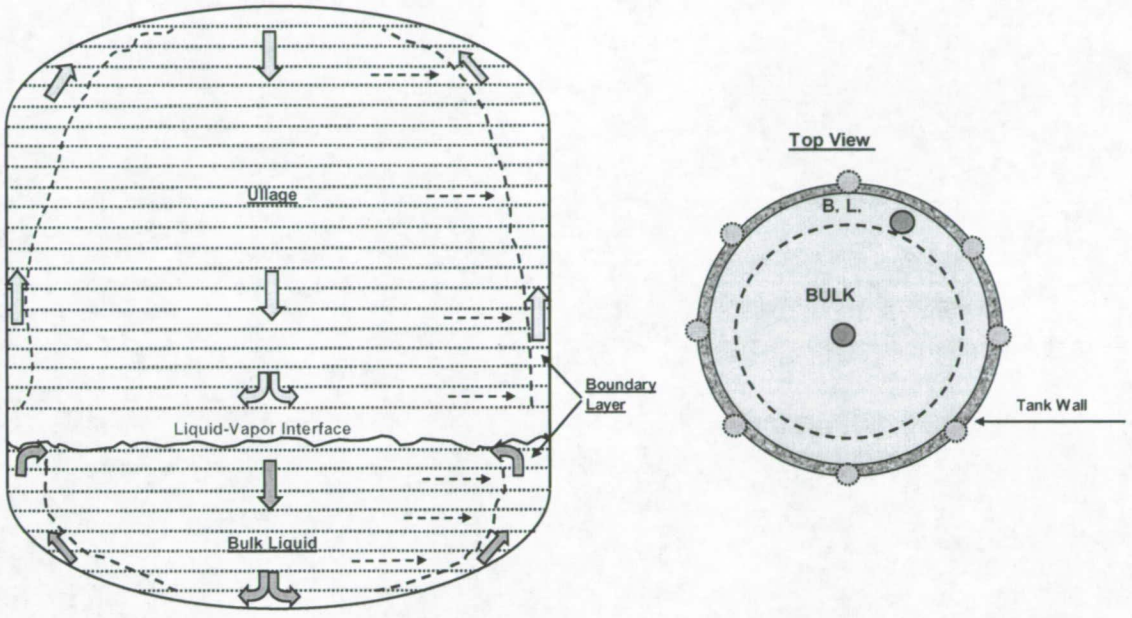

Figure 2: Stratification

Appendix A contains a summary of the equations used to determine convective heat transfer coefficients and boundary layer variables. Characterization of the boundary layer is essential to any attempt to model thermal stratification. The boundary layer provides the mechanism through which warm fluid adjacent to the wall is eventually transported via buoyancy forces to the upper regions of the liquid volume. A boundary layer subroutine provides values for mass flow rates within the boundary layer as well as the local boundary layer thickness. Increasing boundary layer flow in the axial direction is provided for by radial connections between the lumps of the bulk fluid and the lumps representing the boundary layer.

The number of layers the tank is divided into along its axis is at the discretion of the modeler. The number of layers is defined using the register variable 'SLICE'. Excessive number of divisions may not only be unnecessary but may also adversely affect the model's ability to numerically converge. It was decided early on that the axial fidelity (number of layers) of the thermal/structural wall model would equal the axial

fidelity of the thermodynamic fluid model. This was done in an effort to control conditions at the wall immediately local to the fluid. Another consideration, when deciding the axial fidelity of the model, is to compare the amount of fluid mass represented by an individual layer to the desired accuracy when predicting liquid boil-off, etc. In the fluid model, the boundary layer lump may be 'tied' to any number of wall nodes. The fluid model can accommodate any circumferential distribution of wall nodes. In specific applications of this fluid model, the wall of a hydrogen tank has ten nodes along the circumference of each layer (57 three inch layers) and a model of an Oxygen tank had eight nodes along the circumference of each layer (55 two inch thick layers).

Stratification Model

The flow data block of a specific event model is the principle feature distinguishing one event from another. The SINDA/FLUINT model representation of the stratification event consists of a vertical stack of fluid lumps representing the core or 'bulk' fluid. In parallel, is a stack of lumps intended to account for the boundary layer adjacent to the wall. The boundary layer lumps are annular (like a stack of washers) to the lumps representing the bulk fluid. The fluid lumps represent stationary control volumes whose sum equals the total volume of the tank. The thermodynamic status (liquid or vapor) of each lump is determined through a comparison of the liquid depth in the tank to the axial height of a lump and adjacent wall node. Thermal conduction between the fluid lumps is accounted for using FTIEs.

Connectors are network elements used to link lumps together into a network and to provide the means for energy and mass transport between fluid lumps. SINDA provides various types of connectors. 'LOSS' connectors are selected to model the axial mass flow between adjacent bulk fluid lumps and the radial mass flow from the bulk lumps to adjacent boundary layer lumps. 'MFRSET' connector devices are selected to model flow in the axial direction within the boundary layer. MFRSETs are used to transport user defined mass flow rates. Boundary layer flow rates used for the MFRSET connectors are obtained from the 'BLAYER' subroutine. Boundary layer thickness is used to estimate and update the volume of the boundary layer lumps. No mass is transferred, between lumps of different thermodynamic state, except for the TWIN tank at the liquid surface. In a TWIN tank, two distinct tanks are formed from one homogeneous lump at the beginning of the solution, a liquid volume and a vapor volume. The two sub volumes within the TWIN tank are coupled by a superpath. A superpath may be thought of as a set of parallel unnamed MFRSET-like paths, each transporting a different species or perhaps a different phase. Three paths are created between the twinned tanks. The TLIQ subpath is created for the transfer of liquid between the lumps, the TVAP subpath for the transfer of vapor between the lumps and TSPEC subpath to track species diffusion between the tanks.

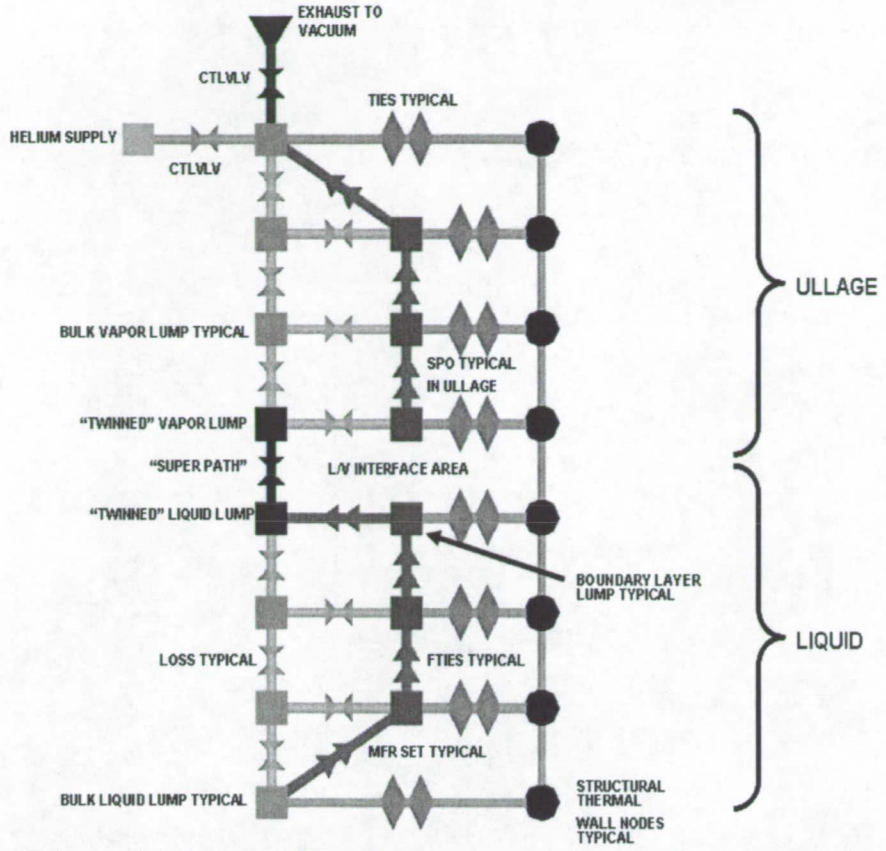

Figure 3: Schematic of SINDA/FLUINT Network Elements for Stratification Modeling

TIEs are generated in the axial direction for each division along the tanks circumference. 'UA' values are updated in appropriate flow logic blocks. Pressurization and venting of the tank is modeled using CTLVLV and UPRVLV (control valve simulations) connectors between the bulk ullage and plenum devices. Plenums are infinite volume SINDA network elements used to provide boundary conditions to an open fluid model. Simple control logic is provided for the opening and closing of these connections corresponding to the desired 'cracking' and 'seat' pressure settings.

Register Data: Registers are used to set initial conditions, track variables, set control flags and determine tank geometries, etc.

Flow Logic: This Block contains most of the logic associated with determining heat transfer conditions at the wall. The following is a high level view of the logic sequence provided.

First, current values of mission dependent variables such as valve operation and gravity (optional input is heat flux or assumed ΔT at the wall) are obtained from the appropriate arrays using the present mission time. Heights of individual layers (wall nodes and adjacent fluid lumps) are determined and the depth of the liquid is updated. Next, conditions for convection at each TIE are determined. The thermodynamic state of the adjacent fluid lump is used to determine conditions used in subroutines which will return values for boiling or natural convection. These values are obtained through the use of successive 'Do Loops' used to navigate within the network axially and circumferentially

in the aft dome, the cylindrical section and finally the forward dome. 'FLOGIC 2' is used to update lump volumes and flow areas.

FLOGIC 2; logic is provided following each time step. FLOGIC 2 provides an opportunity to update the location of the TWIN tank to correspond with the liquid surface. The proper updating of the TWIN tank location is necessary for evaporation and non-equilibrium modeling to occur within the proper liquid layer.

Stratification Modeling Results

Results of the stratification model are graphically illustrated in Figure 4. Figure 4 illustrates the predicted extent to which stratification occurs in three different cryogenic fluids versus time. In this simulation the temperature difference between the wall and the adjacent fluid was a controlled ($\Delta T = 1.0°R$) boundary condition for comparison purposes. The criteria assumed for the existence of stratification was a predicted temperature difference of 0.1 °R between the fluid layer and the remaining bulk liquid temperature. Figure 4 indicates, a) Of the three fluids examined, hydrogen exhibits the strongest tendency for stratification and oxygen the least and b) the amount of stratification is directly related to the level of gravity. These results are as expected. Thermophysical properties of the three fluids are responsible for their relative volatility when subjected to similar boundary heating. Specific properties of interest here are the coefficient of thermal expansion (β), viscosity and specific heat. The influence of gravity is as expected due to its direct influence upon buoyancy forces responsible for circulation of the boundary layer.

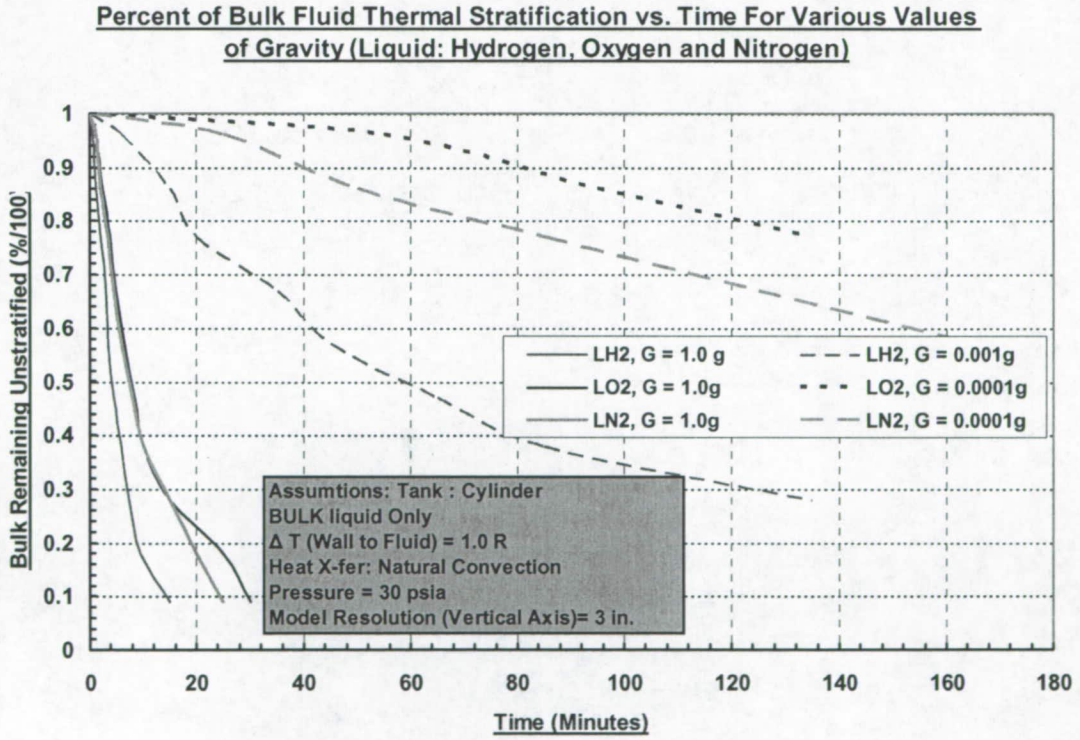

Figure 4: Percent of Bulk Fluid Thermal Stratification vs. Time (Various Values of Gravity)

Rotation Event

The rotation of space vehicles about its longitudinal axis (BBQ roll) is not uncommon. Vehicle roll is often performed for the purpose of providing passive thermal control (PTC). Rotation of a fluid container results in the free surface of the liquid attaining a paraboloid of revolution. The rotation model was constructed to simulate fluid behavior during periods of vehicle rotation. The model is intended to account for the growth of the liquid up the tank wall and to approximate the corresponding change to the liquid-vapor interface area. As the fluid ascends wall space previously opposite of the ullage, the liquid may come in contact with hot wall areas resulting in fluid evaporation and quenching of the wall material.

The fluid network simulates the changing shape of the fluid through the use of TIE connectors. It was decided that the methodology for the fluid network would be to have the lumps maintain their state and alter their TIE to wall nodes as required. This scheme is preferred to the complicated task of tracking the fluid boundary and corresponding thermodynamic states through multiple control volumes.

The rotation event model (Figure 5) and the stratification model are significantly different in the following ways:

1. The rotation model has fewer fluid lumps. Changes to the shape of the fluid can be extreme, especially in a low gravity environment. The change in shape would be complicated to follow with a fluid network methodology similar to the stratification model (i.e. fluid moving through multiple control volumes). The need for large numbers of fluid lumps, such as that used in the stratification model, is reduced due to increased fluid mixing during rotation and low gravity.

2. Fluid lumps used in the rotation model do not change thermodynamic state as the control volumes in the stratified model do. Fluid lumps of the rotation model may change size due to pressure changes, evaporation and condensation.

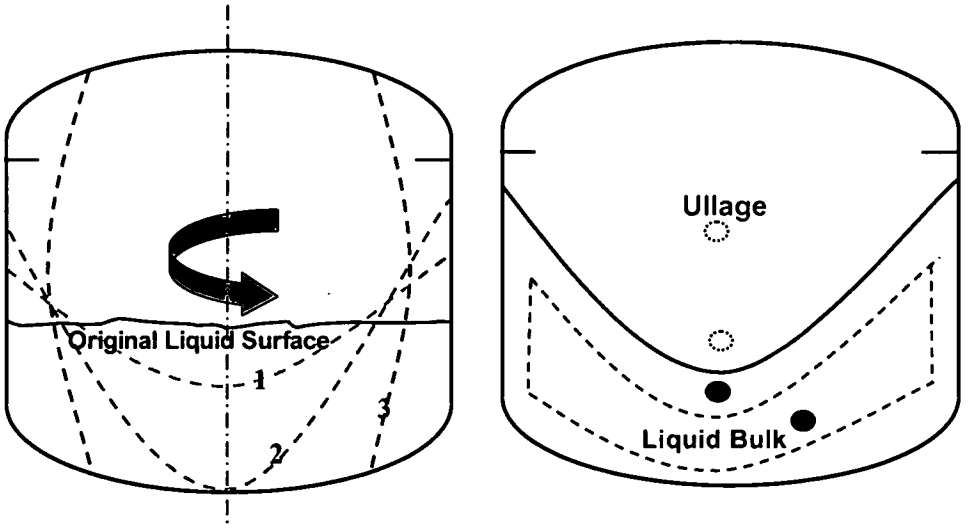

Figure 5: Fluid Rotation

3. Only one liquid lump is in contact with the wall at all times. This liquid lump also provides the liquid-vapor interface and envelopes the bulk liquid lump (Figure 4).

4. The rotation model allows for variable mixing to occur in the liquid and ullage regions of the tank (i.e. between adjacent fluid lumps of the same thermodynamic state). The use of a single lump liquid to wall contact eliminates the need for tracking the location and shape of multiple lumps. This liquid lump creates a 'warm layer' adjacent to the wall and at the liquid-vapor interface. This liquid lump is 'twinned' to allow for non-equilibrium two phase conditions at the liquid-vapor interface. As in the 'stratification' model, a TWIN tank network element is used to accommodate mass transfer between the liquid and the vapor. Heat is conducted between adjacent lumps through the use of FTIE connectors.

Rotation Model

Two lumps represent the liquid contents and one lump represents the vapor contents. A second TWIN vapor lump is created when the liquid lump is twinned (Figure 6). Paths (LOSS connectors) allow for the mass transport between adjacent liquid and adjacent vapor lumps. Four paths allow for the simulation of fluid mixing. Mixing is simulated using two flow paths, MFRSET connectors (one in each direction) between two lumps of the sane thermodynamic state. The extent of the mixing is controlled through the use of register variables 'ulmx' and 'bulkmx'. Venting and pressurization is provided for using 'CTLVLV' devices. This is identical to the method of venting and pressurization control used in the stratification model. FTIE connectors are used to provide heat conduction between adjacent lumps.

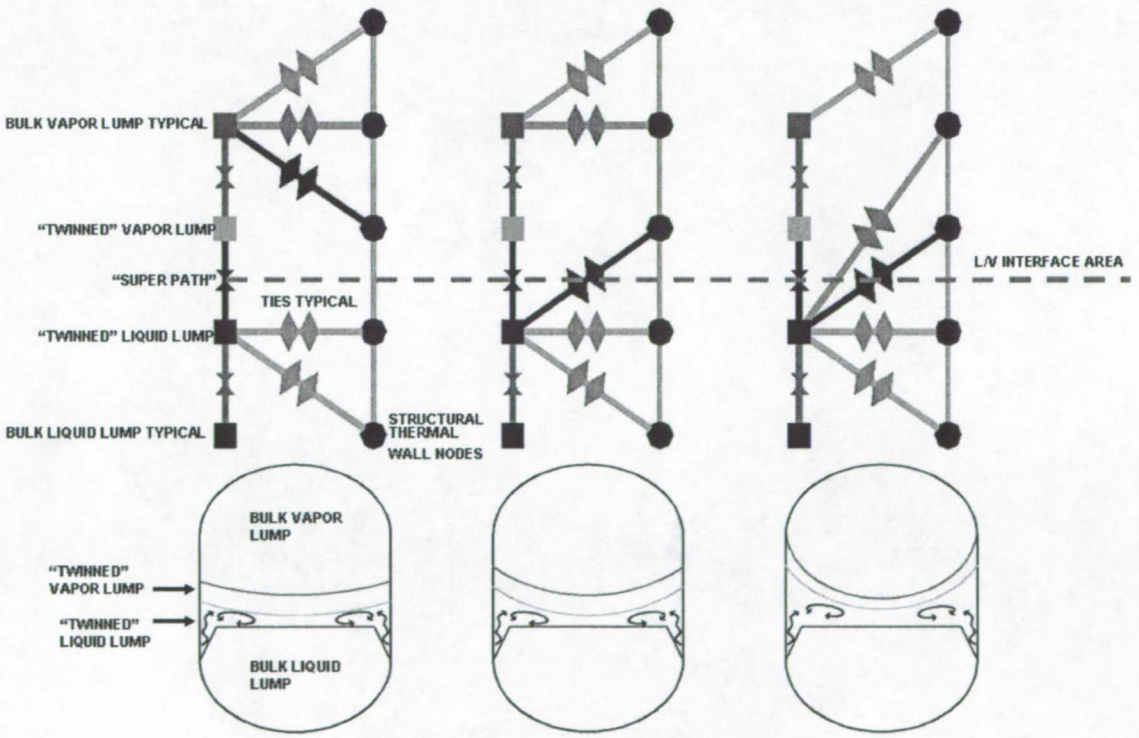

Figure 6: Rotation/Slosh modeling in SINDA/FLUINT via PUTTIE routine

Register Data: There is only one register block in a model.

Flow Logic: This Block contains most of the logic associated with determining heat transfer conditions at the wall. First, mission variables (rate of rotation, vent mode status, etc.) are updated for current mission time. Heights of individual wall nodes are determined. (Remember that the thermal structural heat transfer model is the same for all fluid event models). The height of the liquid and the area of the liquid–vapor interface area are updated from tri-variant storage arrays. These arrays also account for possible migration of the liquid away from the bottom surface (dry wall where there was liquid, line '3' in Figure 5). Inputs to the tri-variant arrays are tank fill percentage, rate of rotation (deg/sec), and gravity ratio (g/g_c). Data in these arrays conform to contributions from CFD simulations.

Successive 'do loops' navigate through the network of wall nodes for each circumferential division in the axial direction. TIEs are assigned to either the liquid 'warm layer' or to the bulk ullage lump based upon the depth of the fluid at the wall compared to the height of the wall nodes.

Rotation Modeling Results

Figure 7 is a composite of three figures from a liquid hydrogen tank simulation. Figure 7 is intended to provide graphical illustrations of essential input assumptions together with selected results for their comparison. Additional assumptions are provided in a text box atop of the bottom figure.

The top figure illustrates the inputs assumed for periods of tank rotation and slosh. The height of the liquid along the tank wall and the area at the liquid-vapor interface directly corresponds to the periods of rotation and the rate of rotation. There are a total of four periods of rotation assumed in the simulation (two at 0.8 deg/sec, and two 1.3 deg/sec). Two slosh events were included in this simulation for comparison to the effects of rotation. Although slosh periods were simulated during periods of no rotation, slosh simulation may also be simultaneous to the rotation simulation. The value of the 'slosh' variable determines the option used to simulate a slosh period. Zero (0) indicates no slosh, one (1) indicates use of the 'zone' option for slosh and two (2) represents periods of time when the 'nodal' option for slosh simulation is being used.

The second figure illustrates the predicted temperatures of the wall at various heights along the tank axis of rotation. Predicted fluid temperatures (warm layer and bulk liquid) are plotted against a separate axis in the figure scaled to fluid temperatures. Note that the bulk temperature exhibited a small yet steady temperature rise. The 'warm' layer exhibits rapid temperature responses influenced by the pressure oscillations occurring in the tank (bottom figure). These pressure oscillations (bottom figure) are most closely associated with the cracking and reseating of the vent control valve. The magnitude of this effect would be expected to vary with the structural mass and the temperature of the wall (latent heat). Note that the change in the slope (psi/sec) of the tank pressure rise associated with the period of the first slosh occurrence (3000 seconds to 3120 seconds) compared to the slope of the previous pressure rise due to nominal conditions. Liquid contents are in contact with increasing wall area and increased areas of contact with the

relatively hot ullage conditions during periods of rotation. Note the decreasing tank pressure following the first rotation period of 1.3 degrees per second (at approximately 5520 seconds). This pressure decline is partially attributable to the presence of cooled wall areas following the rotation period. The following pressure rise is aided by a short (one minute) nodal slosh period.

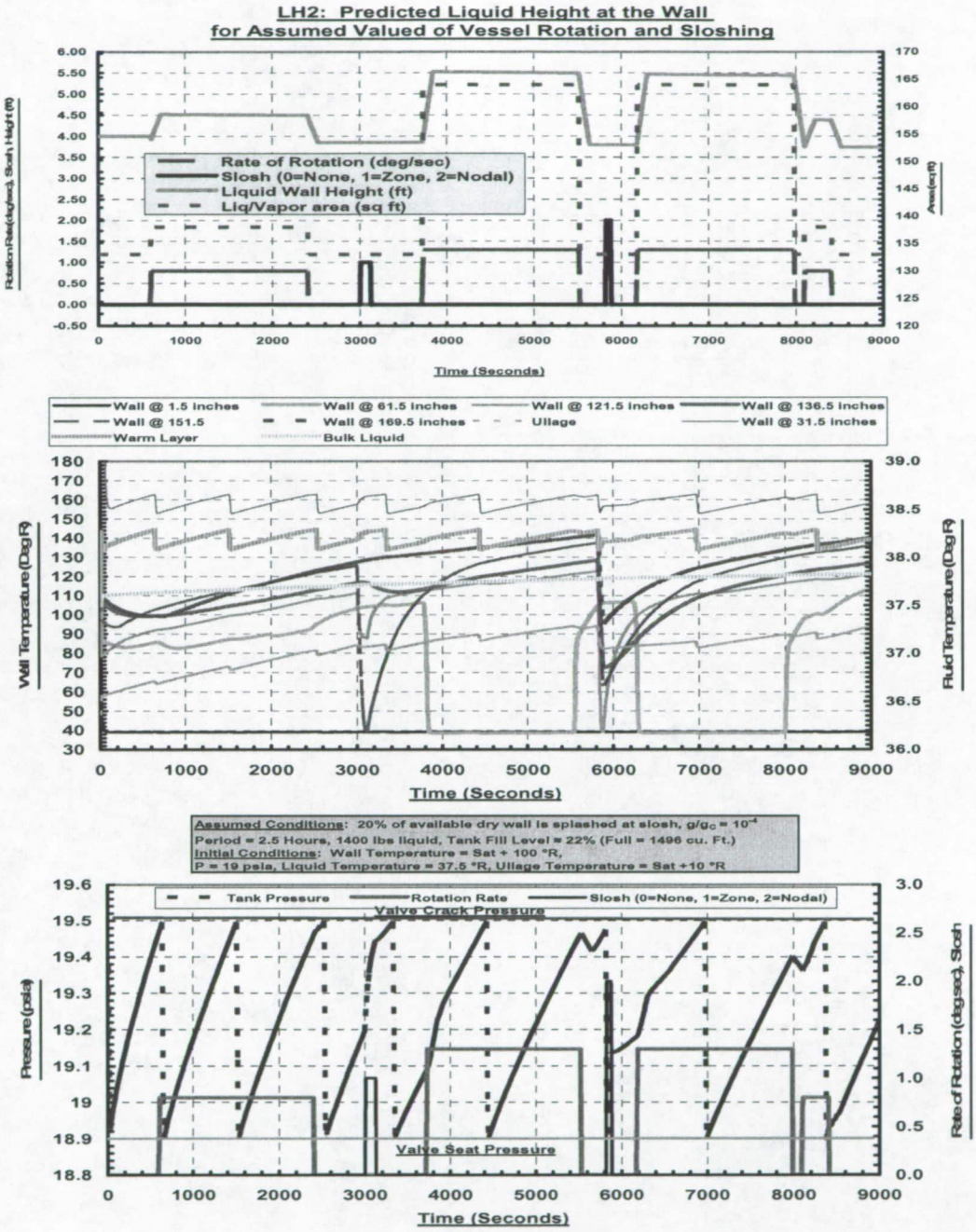

Figure 7: Predicted Influence of Rotation Upon LH2

Slosh Event (See Figure 8)

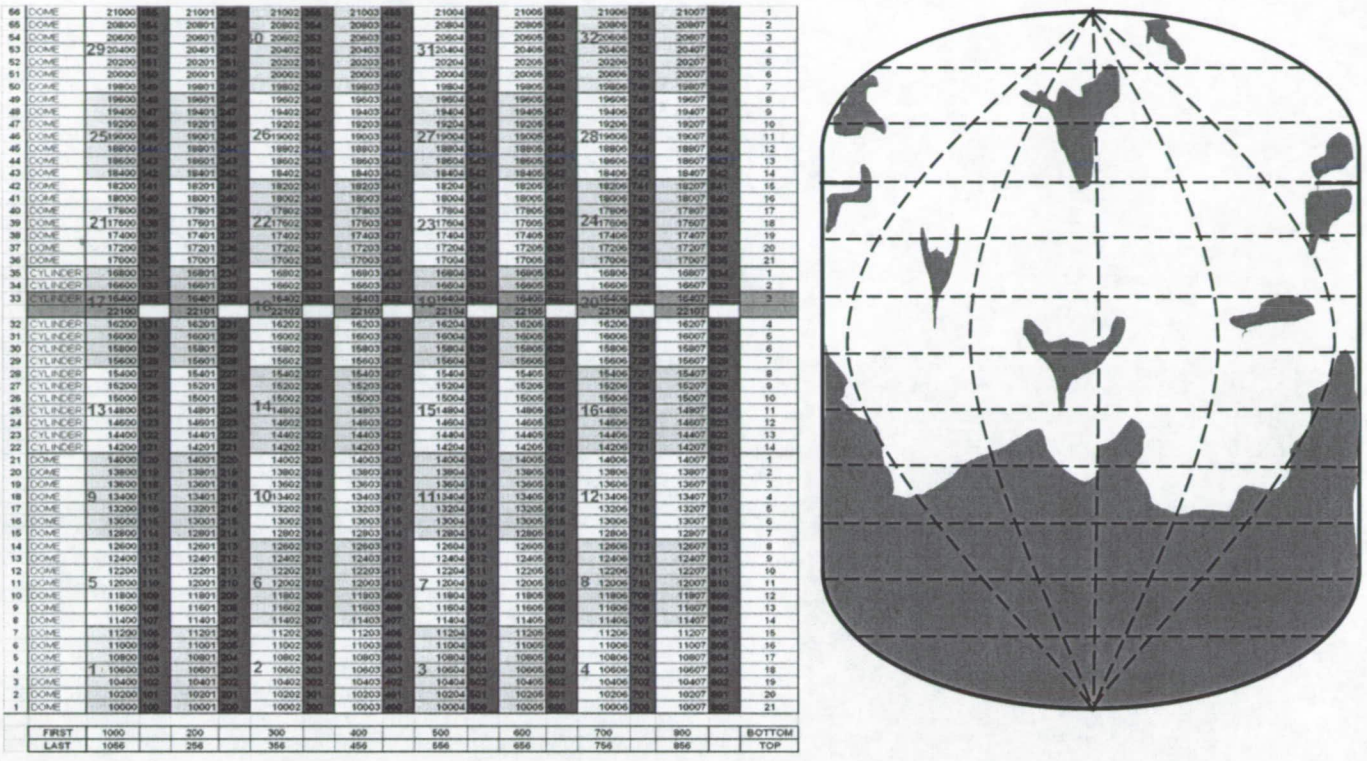

Figure 8: Fluid Slosh – Zone and Individual Nodal Breakdown

Slosh Model

Liquid 'slosh' is a chaotic and short lived event. The fluid network used to model slosh is identical to that of the rotation event. The chaotic nature of the event provides the rationale for not requiring the fluid modeling fidelity required of a stratification model. Neglecting vehicle attitude control and the potential to pass valuable liquid through the venting system; the slosh phenomenon is of interest because of the potential to effect tank pressure and liquid boil off.

Because of the network similarity and the short duration of the event, it is more efficient to call a subroutine for slosh from the rotation model. The modeling of slosh requires the ability to TIE the liquid lumps to the appropriate areas of previously dry wall for the period of the slosh event and then reconnect the TIE to the ullage region at the end of the slosh period. Essentially the modeling of slosh is most similar to the rotation model except for the possible discontinuities in the distribution of wetted wall space during a slosh event. Wall nodes of the surrounding thermal heat transfer model provide a convenient grid upon which areas to be splashed may be identified (Figure 8). The fidelity of the slosh model is to a great extent, dictated by the fidelity of the wall node

distribution. In Figure 8, zones are indicated with large bold numbering and individual wall nodes are indicated with six figure numbers in the background.

Elements: Network elements are identical to Rotation model.

Register Data: There is only one register block in a model.

Flow Logic: A register variable 'SLOSH' is read from an array which is a function of mission time. 'SLOSH' is a numerical key identifying the presence and non presence of sloshing and the slosh option to be used. Zero (0) indicates no slosh, one (1) indicates slosh and the wetting of surfaces identified through the use of zones, and two (2) slosh and the wetting individual wall nodes. This method will allow for any number of slosh events and any combination of slosh options to be used during a mission.

SPLASH Subroutine: The basis of the slosh subroutine is identifying wall to be wetted and the use of the 'PUTTIE' routine. The 'PUTTIE' is a SINDA/FLUINT routine which allows moving of a TIE from one lump to another and/or from one node to another. In this model the TIE is moved between lumps containing vapor and lumps containing liquid. The subroutine for slosh behavior is 'SPLASH'. SPLASH is called if the value of the register variable 'SLOSH' (read from an array) is equal to either '1' or '2' (see Flow Logic above). A double subscripted variable 'STATE' is initially loaded with ones (default of '1' indicates vapor adjacent to the wall). 'STATE' is used to identify individual wall nodes wetted in a slosh event. 'STATE' is dimensioned (double sub-scripted) to represent the number of circumferential and axial divisions of the tank wall model.

If the zone option (option 1) is used to identify wetted areas, then 'singlet' arrays are made available, containing either a zero (0), indicating may be wet but not splashed, or a one (1) indicating the zone is 'splashed'. An array of this type is used to identify the zones to be tied to the liquid lump during an individual slosh period. If there are three slosh periods anticipated for a particular mission, then, three such arrays identifying the splashed zones are used. Each individual location in the singlet array represents the splash status of a zone. A zone is a collection of wall nodes. The use of the zone option allows one to identify groups of wall node to be wetted during a slosh period. The subroutine goes on to test each of the zones for their slosh status. If the zone is to be wetted, then, the wall nodes contained in the respective zone is tied to the fluid lump used for wall heat transfer. Individual nodes represented by a 'wet' zone are also coded as wet through the use of the 'STATE' variable. 'STATE' is later used as the basis for selecting the appropriate model of convection and the use of the correct fluid properties.

If the individual wall node option (option 2) is used to identify areas to be wetted, then, an array of zeroes and ones is used to identify the fluid lump to be tied to. Successive calls, one call for each circumferential division of the wall model, load the contents of that array into the double subscripted variable 'STATE'. Zero (0) indicates wet, one (1) indicates vapor. Individual locations in 'STATE' are tested for their value and TIEs are assigned appropriately to either a liquid or vapor fluid lump.

Following the identification of wetted wall nodes and the reassignment of TIEs to the appropriate fluid lump, STATE is used to determine the appropriate convection subroutine to be used. Because this subroutine is used only when slosh is indicated, a return to the main program will reassign the TIEs to the appropriate fluid lumps, based upon liquid height, upon the completion of the slosh period.

Slosh Modeling Results

The sequence below depicts the node slosh option in which a predetermined set of warm wall nodes are tied to the cold liquid for a certain amount of time and then disconnected. Each frame in Figures 9 and 10 represent snapshots in time. The first frame depicts the fluid and container at a quiescent state. In the second frame, formerly warm nodes adjacent to the ullage volume, are cooled in response to being splashed during a slosh period. Wall space adjacent to the splashed wall nodes are also cooled due to the conduction modeled within the wall. Frame three illustrates the residual effects upon the wall temperatures following the simulated slosh period.

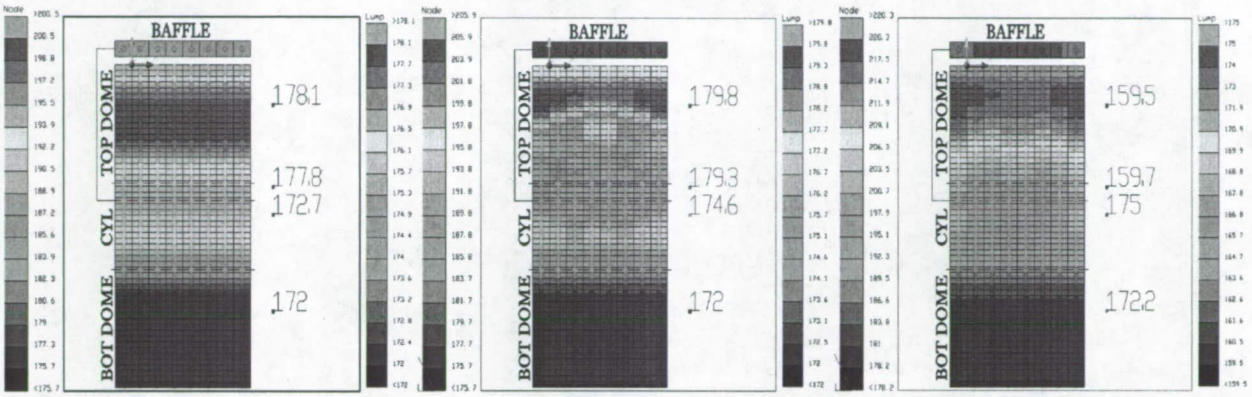

Figure 9: Nodal Slosh Sequence

The sequence below depicts a zone slosh option where a set of node clusters are wetted for a portion of time and then the ties are disconnected and connected back to the ullage vapor. The function of the frame displays are as described above. Note that the use of the zone option may be convenient but lacks the fidelity of the individual node option.

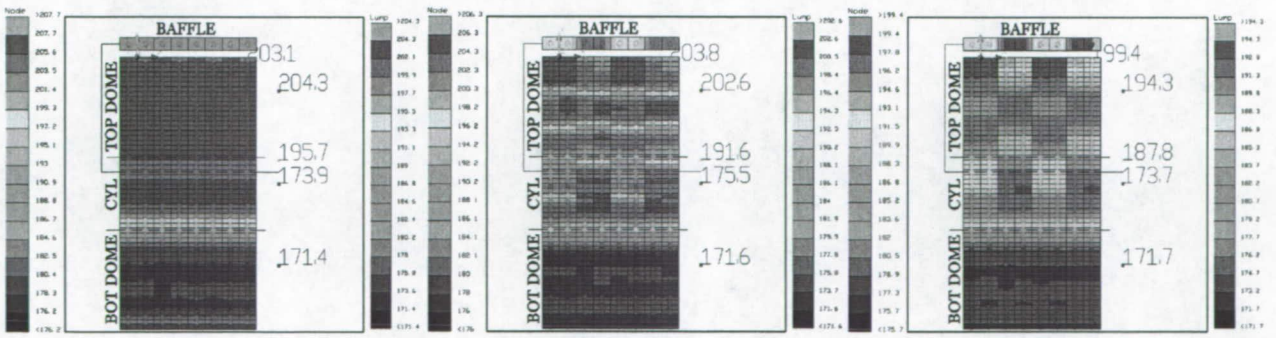

Figure 10: Zone Slosh Sequence

Conclusions

As illustrated within the context of this paper, a tool suite has been successfully developed for use in predicting propellant thermodynamics for upper stage(s) of launch vehicles. The underling development platform is the commercially available SINDA/FLUINT heat transfer/fluid network code. The tool suite consists of "Event Models" (Stratification, Rotation, and Slosh), each of which simulates distinct physical phenomena imposed upon the propellant during flight. The event models run concurrently, with necessary common information being shared via registers within SINDA/FLUINT. Results of the modeling of each event have been successfully demonstrated and presented, along with a description of the integrated modeling approach. This suite will form a foundation for future analytical efforts involving cryogenic upper stages within NASA's Launch Services Program.

Future Work

The current thermodynamic tool suite, presented herein, will be adapted to the current EELV fleet on an as needed basis and can be easily adapted for use on the CLV and CaLV, if desired.

References

1. Cullimore, B.; Ring, S.; Johnson, D.: *SINDA/FLUINT User's Manual*, Version 4.8, C&R Technologies, 2005.
2. Bailey, T.; VandeKoppel, R.; Skartvedt, D.; Jefferson, T.: "Cryogenic Propelllant Stratification Analysis and Test Data Correlation," *AIAA Journal*, Vol. 1, No.7 p. 1657-1659, 1963.
3. Ostrach, S.; and Pradhan, A.: Surface-Tension Induced Convection at Reduced Gravity, *AIAA Journal*, Vol. 16, No. 5, May 1978.
4. Reynolds, W.C.; Saterlee H.M.: "Liquid Propellant Behavior at Low and Zero g," *The Dynamic Behavior of Liquids*, 1965.
5. Ratner, M.; Fumitaka, G.; Kirk, D.; Schallhorn, P.: "Modeling of Upper Stage Cryogenic Propellant Stratification in a Rotating, Reduced Gravity Environment," Thermal and Fluids Analysis Workshop (TFAWs) 2005, Orlando, FL, August 8-12, 2005.
6. Panczak, T.; Ring, S.; Welch, M.; Johnson, D.: *Thermal Desktop User's Manual*, Version 4.8, C&R Technologies, 2005.

Appendix A: Summary of Equations Used In Fluid Subroutines

1. Subroutine 'BLAYER' for boundary layer thickness and mass flow rate;

   A. Boundary layer thickness;
      1) $\delta/x = 3.93\ [(0.952 + Pr) / Gr\ Pr^2]$ (Laminar)

2) $\delta/x = 0.565\ Gr^{-1/10}[(1. + 0.494\ Pr^{2/3}) / Pr^{8/15}]^{1/10}$ (Turbulent)

B. Boundary layer flow rate;

1) $m_{BL} = \bullet W\ (0.0833\ \delta)$ (Laminar)

2) $m_{BL} = \bullet W\ V\ (0.1436\ \delta)$ (Turbulent)

where : V (velocity) = $(\upsilon/x)\ 1.185\ Gr^{1/2}\ [1. / (1. + 0.494\ Pr^{2/3}\ ]^{1/2}$

2. Subroutine 'BOILALL' for boiling heat transfer;

A. Critical Heat Flux;

$Q_{CHF} = 0.18\ \bullet_V\ h_{fg}[\ \bullet\ gg_c\ (\bullet_L - \bullet_V\ ) / \bullet_V^2\ ]^{1/4}\ (\bullet_L / (\bullet_{L+}\ \bullet_V))^{1/2}$

B. Minimum Heat Flux for Film Boiling;

$Q_{Min} = 0.09\ \bullet_V\ h_{fg}[\ \bullet_L\ g\ g_c\ (\bullet_L - \bullet_V\ )\ \bullet_L\ / (\bullet_L + \bullet_V)^2]^{1/4}$

C. Leidenfrost Temperature (i.e. Minimum temperature required for film boiling)

$T_{Leid} = 0.127\ (\bullet_V\ h_{fg}/\ k_V)\ [\ g\ (\bullet_L - \bullet_V\ ) / (\bullet_L + \bullet_V)\ ]^{2/3}\ [\bullet / g\ (\bullet_L - \bullet_V\ )]^{1/2}$

$\cdot[\bullet_V / g\ (\bullet_L - \bullet_V\ )]^{1/3} + T_{Sat}$

D. Departure from film boiling, is a correction to T_{Leid} which often results in excessive temperature. T_{dfb} provides a lower limit based upon Ramilison and Leinhard.

$T_{dfb} = 0.97\ (T_{Crit})\ [0.932 + 0.077\ (T_{Sat} / T_{Crit})^9] + 0.03\ T_{Sat}$

E. Nucleate boiling heat transfer coefficient;

$H = \bullet_L\ (\Delta T/\ h_{fg})^3\ \ (g\ (\bullet_L - \bullet_V\ )/\ g_c\bullet)^{1/2}\ [C_{PL}\ /\ Pr^{1.7}C_{sf}]^3$

F. Film boiling heat transfer coefficients;

1) Vertical plate:

$H = 0.943\ [(k^3_V\ g\ \bullet_V\ \ (\bullet_L - \bullet_V\ )\ h_{fg}\ )/(x\ \bullet_V\ (T_{Wall} - T_{Sat}))]\ [3600]^{1/2}$

2) Horizontal plate:

$H = 0.425\ [(k_V\ g\ \bullet_V\ \ (\bullet_L - \bullet_V\ )\ h_{fg}\ )\ /\ (x\ \bullet_V\ (T_{Wall} - T_{Sat}))]$

$\bullet[(\bullet\ gc/g\ (\bullet_L - \bullet_V\ ))]^{1/2}$

3. Natural Convection;

A. Curve interpolation of Log Ra vs Log Nu $[\ 10^{-1} < Ra < 10^4]$

then, $H = Nu\ k\ /\ x$

Vertical correlations:

B. $H = 0.59\ (Ra^{0.24}\ k\ /\ x)$ $[\ 10^4 < Ra < 10^9]$

C. $H = 0.129\ (Ra^{0.33}\ k\ /\ x)$ $[\ 10^9 < Ra < 10^{14}]$

Horizontal correlations

D. $H = 0.54\ (Ra^{0.24}\ k / x)$ $[10^5 < Ra < 2x10^9]$

E. $H = 0.14\ (Ra^{0.33}\ k / x)$ $[2x10^9 < Ra < 3x10^{10}]$

4. Molecular Diffusion:

Ficks Law: $$D_{AB} = \frac{0.02195T^{3/2}\left(\frac{1}{M_A}+\frac{1}{M_B}\right)^{1/2}}{P(r_{AB})^2[f(\theta)]}$$

Where:

T = Ullage temperature [°F]
D_{AB} = Diffusion coefficient [ft^2/hr]
C_{sf} = Boiling coefficient due to surface roughness
C_{PL} = Specific heat [Btu/lbm-°F]
$f(\theta)$ = collision function for the two species
g = local gravitational acceleration [ft/sec^2]
H = Convective heat transfer coefficient [Btu/hr-ft^2-°F]
h_{fg} = heat of vaporization [Btu/lbm]
k = Thermal Conductivity [Btu/hr-ft-°F]
M_A, M_B = Molecular weights of the respective components
m_{BL} = Boundary layer mass flow rate [lbs/hr]
P = Absolute pressure [psia]
r_{AB} = molecular separation factor
W = circumference [ft]
x = characteristic dimension [ft]

Gr = Grashof Number

Nu = Nusselt Number
Pr = Prandtl Number
Ra = Rayleigh Number

Δ = Boundary layer thickness [ft]
υ = kinematic vixcosity [ft^2/hr]
• = surface tension [lb/ft]
• = Dynamic Viscosity [lbm/ft-hr]

Subscripts:

L = Liquid
V = Vapor
C = Constant of proportionality [gravity: lbm-ft/lbf-sec^2]
AB = Species 'A' and species 'B'