Paper Number 2000-XX-YYYY

Steady-State and Transient Loop Heat Pipe Modeling

Brent Cullimore
Jane Baumann
C&R Technologies, Inc.

ABSTRACT

The NASA-standard thermohydraulic analyzer, SINDA/FLUINT (Ref 1), has been used to model various aspects of loop heat pipe* (LHP) operation for more than 12 years. Indeed, this code has many features that were specifically designed for just such specialized tasks, and is unique in this respect. Furthermore, SINDA is commonly used at the vehicle (integration) level, has a large user base both inside and outside the aerospace industry, has several graphical user interfaces, preprocessors, postprocessors, has strong links to CAD and structural tools, and has built-in optimization, data correlation, parametric analysis, reliability estimation, and robust design tools.

Nonetheless, the LHP community tends to ignore these capabilities, yearning instead for “simpler” methods. However, simple methods cannot meet the challenging needs of LHP modeling such as transient start-up and noncondensible gas (NCG) effects, are often hardware-specific or proprietary, or cannot be used in a vehicle-level analysis.

There are many reasons for this hesitancy to use SINDA/FLUINT as it was intended. First, hardware developers tend to be less versed in analytic methods than the user community they serve. Second, there are political hurdles, such as the fact that ESA contractors are required to use ESA sponsored software. Third, the state-of-the-art in LHPs is not so advanced that the analysts can be ignorant of the complex two-phase thermohydraulic and thermodynamic processes and phenomena involved, and unfortunately most thermal analysts are accustomed only to “dry” thermal control (radiation, conduction, etc.).

Fourth, the general-purpose and complete nature of SINDA/FLUINT tends to make it intimidating, especially in light of the third reason listed above. SINDA/FLUINT is not designed strictly for LHPs or even for LHP-like systems; it has been used for everything from nuclear reactor cooling to dynamic models of human hearts and tracheae. The user’s manuals and standard training classes† rarely mention capillary phenomena because only a fraction of SINDA/FLUINT’s users are thus inclined. It is to address this fourth reason that this paper has been written, since the authors can do little to redress the first three problems.

This paper summarizes the available modeling capabilities applicable to various LHP design and simulation tasks. *Knowledge of LHPs is assumed.*

INTRODUCTION: THERMAL/FLUID NETWORKS

SINDA/FLUINT is the NASA-standard heat transfer and fluid flow analyzer for thermal control systems. Because of its general formulation, it is also used in other aerospace specialties such as environmental control (ECLSS) and liquid propulsion, and in terrestrial industries such as the electronics packaging, automotive, refrigeration, and power generation industries.

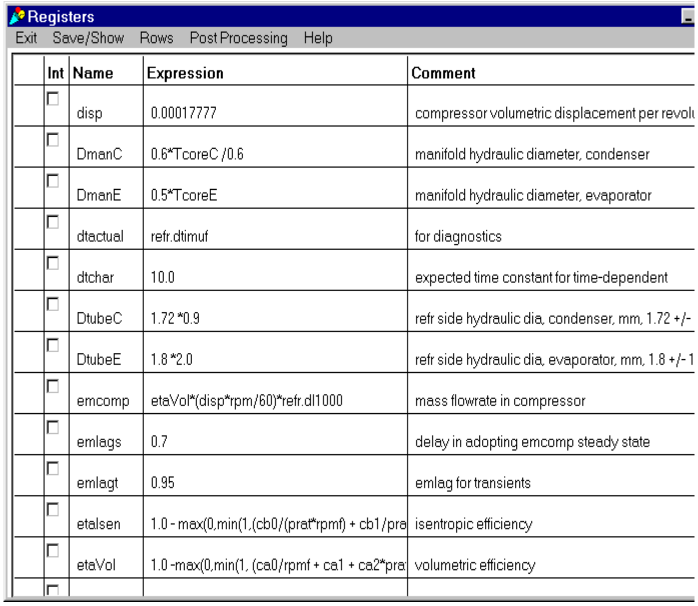
SINDA/FLUINT is used to design and simulate thermal/fluid systems that can be represented in networks corresponding to finite difference, finite element, and/or lumped parameter equations. In addition to conduction, convection, and radiation heat transfer, the program can model steady or unsteady single- and two-phase flow networks, including nonreacting mixtures and nonequilibrium phenomena.

A built-in spreadsheet enables the user to define custom (and perhaps interrelated) variables (Figure 1). The user can also define complex self-resolving interrelationships between inputs, and also between inputs and outputs. This spreadsheet allows rapid and consistent model changes, minimizes the need for user logic, and makes parametric and sensitivity studies trivially easy to perform. Top-level modules automate design, optimization, test data correlation, reliability estimation, and robust design (reliability-

---

* Although LHPs will be used to demonstrate various modeling capabilities, most of these capabilities are equally well applied to capillary pumped loops (CPLs).

† Classes in capillary modeling *are* taught, albeit rarely because of their specialized nature.



	Int	Name	Expression	Comment
	☐	disp	0.00017777	compressor volumetric displacement per revolu
	☐	DmanC	0.6*TcoreC /0.6	manifold hydraulic diameter, condenser
	☐	DmanE	0.5*TcoreE	manifold hydraulic diameter, evaporator
	☐	dtactual	refr.dtimuf	for diagnostics
	☐	dtchar	10.0	expected time constant for time-dependent
	☐	DtubeC	1.72 *0.9	refr side hydraulic dia, condenser, mm, 1.72 +/-
	☐	DtubeE	1.8 *2.0	refr side hydraulic dia, evaporator, mm, 1.8 +/- 1
	☐	emcomp	etaVol*(disp*rpm/60)*refr.dl1000	mass flowrate in compressor
	☐	emlags	0.7	delay in adopting emcomp steady state
	☐	emlagt	0.95	emlag for transients
	☐	etalsen	1.0 - max(0,min(1,(cb0/(prat*rpmf) + cb1/pra	isentropic efficiency
	☐	etaVol	1.0 -max(0,min(1, (ca0/rpmf + ca1 + ca2*pra	volumetric efficiency
	☐			

Figure 1: Part of the Built-in Spreadsheet: User-defined Registers

based optimization) tasks, far exceeding the capabilities of traditional steady and transient analyses.

Concurrent developments have made these features more accessible. C&R's Sinaps*Plus*® is a complete nongeometric (circuit sketchpad) pre- and postprocessor for SINDA/FLUINT. C&R's Thermal Desktop® (with the optional RadCAD® radiation analyzer) is a geometric (CAD/FEM/FDM) interface that brings traditional thermal modeling practices into a concurrent engineering environment. A freely distributed plotting program is also available: EZ-XY™.

SINDA

SINDA uses a thermal network approach, breaking a problem down into points at which energy is conserved (*nodes*), and into the paths (*conductors*) through which these points exchange energy via radiation and conduction. While often applied as a lumped-parameter modeling tool, the program can also be used to solve the finite difference (FDM) or finite element (FEM) equations for conduction in appropriately meshed shells or solids. In Thermal Desktop, for example, one can employ finite difference, finite element, and arbitrary (lumped parameter) nodes all within the same model.

An important improvement over ancestral versions of SINDA is the inclusion of submodels, which enable analysts to subdivide a large network of nodes and conductors into collections of subnetworks consisting of nodes, conductors, or both. Submodels represent a convenient means of combining separately developed models, each with its own control variables, customization logic, solution method, and perhaps conflicting node and conductor numbering schemes. More often, they are simply used to improve the organization and legibility of the model, or to perform high-level simulation manipulations such as dynamically swapping sets of boundary conditions, evaluating alternate designs or components, or simulating variable configurations.

Solutions may be performed in single- or double-precision without any model or logic changes. Also, either iterative or simultaneous (optimally reordered sparse matrix) solutions may be used in steady-state or transient analyses. SINDA/FLUINT provides a powerful means for creating highly customized solution schemes by permitting the user to vary the underlying methods on a submodel-by-submodel basis.

FLUINT

To answer the need to model two-phase fluid systems and to replace the cumbersome and limited "one-way conductor" methods employed by ancestral versions of SINDA for fluid flow simulation, FLUINT development was initiated by NASA in the 1980's as a major expansion of SINDA. All major development has been completed, providing unmatched thermohydraulic analysis capability. Thermal and fluid models may be used alone or together to solve conjugate heat transfer problems as typically found in thermal control, propulsion, and energy systems.

FLUINT introduced a new type of submodel composed of network elements, *lumps* and *paths*, which are analogous to traditional thermal nodes and conductors, but which are much more suited to fluid system modeling. Unlike thermal networks, fluid networks are able to simultaneously conserve mass and momentum as well as energy.

Lumps are subdivided into *tanks* (control volumes), *junctions* (volumeless conservation points, instantaneous control volumes), and *plena* (boundary states). Paths are subdivided into *tubes* (inertial ducts), or *connectors* (instantaneous flow passages including short ducts [*STUBE* connectors], valves, etc.).

In addition to lumps and paths, there are three additional fluid network elements: *ties*, *fties*, and *ifaces*. Ties represent heat transfer between the fluid and the wall (i.e., between FLUINT and SINDA). Fties or "fluid ties" represent heat transfer within the fluid itself. Ifaces or "interface elements" represent moving boundaries between adjacent control volumes.

Paralleling SINDA while at the same time extending the SINDA design philosophy, FLUINT models can be constructed that employ fully transient thermohydraulic solutions (using tanks and tubes), or that perform pseudo-steady transient solutions (neglecting perhaps inertial effects and other mass and energy storage terms using junctions and STUBE connectors), or that employ both techniques at once. In other words, the engineer has the ability to approximate or idealize where possible, and to focus computational resources where necessary. Like SINDA, full access is provided in logic and in spreadsheet relationships not only to the basic modeling parameters (dimensions, properties, loss factors, etc.), but also to derived or abstract solution parameters (e.g., the exponent on flow rate of the friction coefficient), and to underlying correlations for heat transfer, pressure drop, etc.

Although the user can build models of custom parts and control systems, prepackaged tools are provided for modeling common components such as pipes, pumps, valves, filters, accumulators, etc. Table 1 presents the overall organization of SINDA/FLUINT modeling tools.

The remainder of this paper describes additional features in detail, with example usage based on experience modeling LHPs.

BASIC TWO-PHASE FLOW OPTIONS: CONDENSER AND TRANSPORT LINE MODELING

Two-phase flow is by default homogeneous (equal liquid and gas velocities) and in phasic equilibrium (perfectly mixed: equal temperatures and pressures between phases). Also by default, flow regimes are predicted based on local flow characteristics, orientation with respect to body forces, etc. These regimes are used for pressure drop calculations and also for the more advanced options described later. However, the user can select from several built-in two phase pressure drop options, or can even add their own. Similarly, there exist default correlations for boiling and condensation.

In an LHP, transport lines and condensers with circular cross sections (i.e., pipes) are normally used, and the flow within the those lines is usually annular or slug flow. The default correlations work well under these conditions. If an annular cross section is used, alternate heat transfer correlations should be applied. However, in practise most users simply apply an augmentation factor to the default correlations.*

FLUINT offers *duct macros*, which are convenient means of specifying pipes and other flow passages in which heat transfer might occur, causing large axial changes in heat transfer coefficient and density (and therefore spatial acceleration) in the case of two-phase flow.

Fortunately, these simple tools are often all that is needed to model even complex LHP condensers, including distribution effects between parallel lines, asymmetric sink conditions, etc. Usually, the complexity in models of LHP condensers is not related to thermohydraulics, but rather to the detail required in the thermal-structural model and the environmental model. This underscores the importance of seamless integration with system-level analyses using SINDA and perhaps Thermal Desktop or a similar radiation and orbital environment analyzer.

A word of caution: to generate correct predictions for an LHP, the overall pressure drop through the system must be accurately predicted, as well as seemingly secondary effects such heat exchange with between the environment and the transport lines and compensation chamber. Again, this causes extra resolution to be required in the transport lines and condenser, contributing to model size but not to modeling complexity.

FLUID MIXTURES: NONCONDENSIBLE GASES

Although most analyses require only a single (pure) volatile working fluid such as ammonia, water, propylene, or propane,† the user can add up to 25 nonvolatile liquids (i.e., oils) and noncondensible gases (NCGs) to the working fluid mixture. Masses of each species are conserved.

Normally, the partial pressure offset of an NCG is the greatest effect in a steady LHP, and this offset can be of critical importance during transient start-up. In these cases, the gas can be analytically “injected” into the compensation chamber and neglected elsewhere in the loop.

However, it is also possible to model the effects of NCG and oils on the condenser, including degraded heat transfer due to mechanisms including saturation temperature reduction, liquid film blockage, and diffusion-limited condensation. All of these effects are analyzed by default when mixtures are modeled (Figure 2).

Optionally, the analyst can model dissolution and evolution of noncondensible gases into and out of liquid phases. Equilibrium solubilities of binary solvent/solute pairs may be defined using a variety of rules (e.g., Henry’s, Raoult’s) and coefficients (e.g., Ostwald, tables of mass or mole fractions). By default the code will estimate mass transfer coefficients and interfacial surface areas using knowledge of the flow regime and the interfacial heat transfer coefficients. The user can override or augment this default system, or use it to scale the results as needed to quantify uncertainties or to correlate the model to available test data.

The dissolution and evolution of gases can be neglected in most LHP analyses. However, inclusion of these effects can be important in assessing the movement of NCGs within the LHP during transients, or for fine assessments of the impact of end-of-life gas tolerance. For example, the analyst could assess the utility (or futility!) of using a capillary gas trap at the exit of the condenser by predicting the fraction of gas that will have had time to dissolve into the condensate during passage through the condenser. More often, the analyst may wish to take advantage of the fact that not all generated gas will be in the vapor phase. However, such an “advantage” should perhaps be discarded as

---

* In an realistic application, the bottleneck in heat transport from a condenser to the sink is usually radiation, followed perhaps by contact or bonding conductances: uncertainties in film coefficients are usually secondary or even tertiary considerations.

† All of these fluids are built into FLUINT. Additional properties are freely available for cryogens, alcohols, hydrocarbons, refrigerants (including R134a), etc.

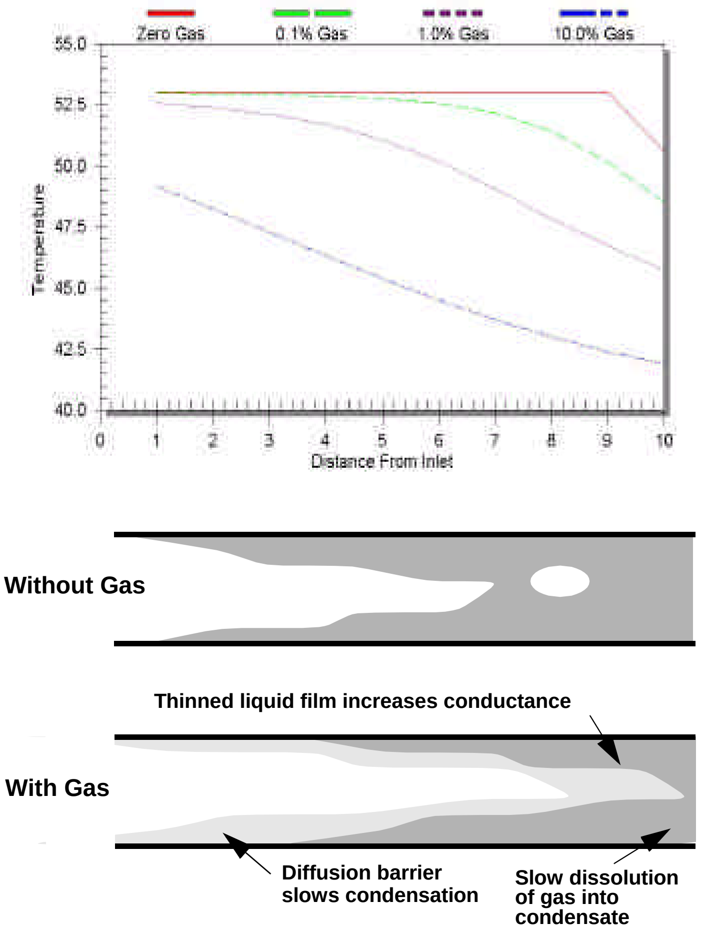


Figure 2: Noncondensible Gas in a Condenser

nonconservative for steady-state performance assessments, and should be applied only as needed to correlate with test data.

ADVANCED TWO-PHASE OPTIONS

SLIP FLOW (NONHOMOGENEOUS FLOW)

The simplifying assumption of homogenous flow is almost always adequate in LHPs, but if desired the momentum equations for liquid and vapor can be solved separately by selecting slip flow: by letting liquid and vapor/gas velocities differ.

Slip flow is easily invoked by simply *twinning* the tubes or STUBE connectors representing the flow passages. This creates two linked paths that together model the flow passage, taking into account wall friction apportionment, accelerations associated with phase change, added (or virtual) mass, and most importantly interphase shear: the rubbing of liquid against vapor. Although users can adjust or inspect any of the above calculations, they usually simply rely on the default methods based on flow regime predictions.

Slip flow modeling is usually only required when (1) void fraction estimations are critical, (2) transient motions of the liquid (perhaps due to vehicle motions) phase are important, or (3) nonequilibrium flow is elected, as described next.

NONEQUILIBRIUM PHASES

Under steady conditions, the liquid and vapor within the LHP compensation chamber are at the same temperature and pressure. However, under transient conditions (especially start-up), cold liquid can fill the compensation chamber while at the same time compressing and therefore heating the vapor. Even though the chamber will eventually drop to a lower pressure than the initial value, there can exist a transient excursion above this initial pressure due to nonequilibrium effects. Worse, if the chamber is assumed to be perfectly mixed, the model may overreact to such transient movements of liquid, leading to artificial responses and perhaps artificial instabilities.

The ability to model temperature and pressure differences between liquid and gas phases within quasi-stagnant control volumes has always been present in FLUINT, although the robustness of these previous solutions was increased tremendously by the addition of ifaces a few years ago (Ref 2). Recently, the ability to twin tanks has been added, replacing the earlier nonequilibrium capabilities and at the same time extending them to encompass flow within pipes as well as quasi-stagnant volumes. Twinned tanks, like the analogous twinned paths used to model slip flow, jointly share responsibility for modeling a single control volume, using separate equations to conserve mass and energy in each phase.

Thus twinned tanks can be used to model nonequilibrium effects within the compensation chamber during severe transients such as start-up, condenser quenching, etc. Otherwise, a single homogeneous (perfectly mixed) tank is usually adequate, and a plenum (infinite source or sink at constant pressure) is also frequently used to model the compensation chamber.

PHASE AND SPECIES SUCTION OPTIONS

One of the simplest options applicable to LHP modeling is the ability to specify that a path “sees” only one phase or one species (if a mixture is used). For example, a path can be made to extract only liquid from an upstream two-phase lump (presuming that liquid is available and that any inflowing vapor can be either accumulated or rerouted or condensed).

Phase suction can be used to model capillary traps and capillary flow regulators, and has other uses in the detailed modeling of components such as compensation chambers, liquid bayonets, and secondary wicks. Species suction can be used to model gas getters.

However, casual users need only note that phase suction options are used internally by many of the capillary modeling options available in FLUINT, such as those described next.

Table 1: SINDA/FLUINT Hierachy of Modeling Options

- Thermal/Fluid Model
  - Registers, Expressions, and Spreadsheet Relationships
  - Concurrently Executed User Logic
  - Thermal Submodels
    - Nodes
      - Diffusion (finite capacitance)
        - Temperature-varying
        - Time-varying
      - Arithmetic (massless: instantaneous)
      - Boundary (constant temp.)
      - Heater (constant temp., returns power)
    - Conductors
      - Linear (conduction, advection)
        - Temperature-varying
        - Time-varying
      - Radiation
        - Temperature-varying
        - Time-varying
    - Sources
        - Temperature-varying
        - Time-varying
  - Fluid Submodels
    - Lumps
      - Tanks (finite volume)
        - Twinned tanks (nonequilibrium modeling)
      - Junctions (zero volume: instantaneous)
      - Plena (constant temperature, pressure)
    - Paths
      - Tubes (finite inertia)
        - twinned tubes (slip flow)
      - Connectors (zero inertia: instantaneous)
        - short tubes (STUBEs)
          - twinned STUBEs (slip flow)
        - valves
          - check valves, control valves
          - pressure regulating valves
        - K-factor losses, bidirectional or not
        - pumps, fixed or variable speed
        - constant mass or volumetric flow rate
        - capillary elements (CAPILs)
    - Ties (heat transfer)
      - user-input conductance
      - program-calculation (convection) conductance
    - Duct macros (subdivided pipelines)
    - Capillary evaporator-pumps (CAPPMP macros)
    - Ifaces (control volume interfaces), with or without inertia
      - flat (zero pressure difference)
      - offset (finite pressure difference)
      - spring (i.e., bellows, etc.)
      - spherical bubble
      - wick (liquid-vapor interface in porous structure)
    - Fties (fluid-to-fluid ties)
      - axial in a duct
      - user-input conductance
      - constant heat rate
    - Auxiliary Utilities
      - choked flow detection and modeling
      - waterhammer and acoustic wave modeling
      - compressors
- Solutions
    - Steady-state
    - Transient
    - Goal Seeking
    - Design Optimization
    - Test Data Correlation
    - Reliability Estimation
    - Robust Design

ADIABATIC CAPILLARY WICK FLOW ELEMENT

A special kind of connector (the inertialess flow passages) is the CAPIL connector. A CAPIL connector is completely defined by an effective 2D capillary radius and a capillary flow conductance (calculated based on shape, size, and permeability for a homogenous wick).

A CAPIL connector refuses to permit vapor to pass, at least until its capillary limit has been exceeded. Otherwise, it represents a laminar flow loss: the pressure drop is proportional to flow rate.

CAPIL connectors, like any path, are adiabatic: heat can only be added or subtracted at the lumps at either end of the CAPIL connector. Therefore, they cannot perform capillary pumping: that is the realm of the CAPPMP macro described in the next section.

CAPIL connectors, however, are used to represent any passage that is small enough that the passage of vapor bubbles is impeded: grooves, wicks, slots, filters, tubules, etc. In LHP modeling, they are convenient to use to represent condenser flow control devices and gas traps, but these components are rarely used in the LHP design. More commonly, CAPIL connectors are used to represent secondary wicks and leakage (nonpumping) paths across the primary wick, or weak spots in the primary wick.

HEATED CAPILLARY WICK FLOW ELEMENT

A CAPPMP macro is very similar to a CAPIL connector, with one important difference: if heat is added to it, it can perform capillary pumping. It is called a “macro” because it consists of multiple FLUINT network elements, but it can basically be thought of as a CAPIL with a junction in the middle. The junction is in the middle *mathematically*, although *physically* heat is usually added to the liquid/vapor interface that is normally located on the vapor side of the CAPPMP macro.

The CAPPMP macro is normally used to represent vaporization at the primary wick. The user may either add constant flux to the CAPPMP, or link it to a SINDA node representing the evaporator wall.

A common misconception is that a single CAPPMP macro should be used to model the entire evaporator. This is not necessary; a CAPPMP macro is merely a building block like other FLUINT network elements. Therefore, multiple CAPPMP macros could be used in parallel to model axial or circumferential gradients within the evaporator. Multiple parallel CAPPMP macros (and perhaps CAPIL connectors) can also be used to take into account the fact that real wicks contain a distribution of pore sizes, and that some portions of the wick can be deprimed while others are still wetted and pumping.

Again distinguishing a CAPPMP macros from a model of an evaporator, note that CAPPMP macros can be used if needed to model capillary vaporization wherever it occurs, such as within the secondary wick of a heated compensation chamber.

LIQUID-VAPOR INTERFACES WITHIN WICKS

CAPILs and CAPPMPs model the flows and pressure drops associated with capillary structures, but are simple one-dimensional building blocks. Combining them with ifaces provides even more modeling power: the ability to include motions of the liquid-vapor interface as needed for detailed transients including oscillations and start-up.

FLUINT *paths* are used to represent mass transport from *lump* to *lump* based on pressure differences. FLUINT ifaces ("interfaces") are used to represent motion of the boundaries between *tanks* (the finite volume subset of lumps) based on pressure differences. The most common type of iface is the FLAT iface, which maintains a constant pressure between two tanks. FLAT ifaces are normally used between the liquid and vapor tanks of twinned (non-equilibrium) tanks in the absence of capillary structures, hence the name: a flat liquid-vapor interface has no curvature and therefore there is no pressure difference between the two phases.* Ifaces have optional inertial terms in case the boundary is not massless and must instead be accelerated and decelerated as it moves.

One type of iface is especially important to LHP modeling: the WICK iface. As its name implies, the WICK iface is designed to model the one-dimensional motions of a liquid-vapor interface, including tracking the liquid-vapor front in relation to the wick surface. The inertia of the interface may be calculated based on the mass of the liquid within the wick.

The user normally links the WICK iface directly to a parallel CAPIL or CAPPMP. This has several advantages. First, the WICK iface will then automatically share data (e.g., the capillary radius) with the capillary element. Second, the WICK iface will be able to take into account the pressure drop within the wick itself.

WICK ifaces are necessary when modeling instabilities or self-induced oscillations within the LHP. However, they are a "must" for any high fidelity model employing tanks (control volumes) to model the liquid core and the vapor grooves. Without them, the liquid-vapor interface becomes rigid, and this is a harsh assumption causing unrealistic spikes and notches in the transient pressure difference across the wick. In other words, an active (primed) CAPPMP or CAPIL represents a mathematical discontinuity: the flows in such devices are independent of pressure drop. This discontinuity is eliminated using a WICK iface: the liquid and vapor control volumes may push and pull against each other because of the introduction of a responsive boundary between them.

BODY FORCES

Although body forces (including vehicle motions) have been mentioned as having an effect on flow regime predictions, their primary effect in an LHP is simply a hydrostatic gradient within the liquid. The gravitational pressure drop is very important for LHPs, both in predicting adequate capillarity and also because it affects performance: a tilted LHP does not have the same overall conductance as a horizontal LHP.

Modeling body forces, vehicle accelerations (and even vibrations) is straightforward: the user specifies the coordinate location of each lump in the LHP model in one, two, or three axes. The global acceleration (usually gravity) is defined as a vector with one, two, or three components. Lump locations and acceleration vectors may vary *during* a single run, but more commonly the user simply "tilts" the LHP between runs by changing the acceleration vector.

WICK BACK CONDUCTION MODELING

Because the liquid moves slowly in an LHP, and because the wicks are usually made of metal, heat conducts backwards from vapor to liquid (against the flow). This effect is usually called "back conduction."

Modeling back conduction is both easy and critical. In FLUINT, an FTIE may be placed between liquid and vapor lumps across the primary wick.† The conductance of this "USER" (user-defined) FTIE can be calculated using auxiliary routines.

Preferably, the conductance of a wetted wick is known from test data. Lacking test data, an auxiliary routine is available that contains various correlations (sintered wicks) and limiting cases (parallel upper limit, series lower limit).

Another routine is available to perform a usually minor correction on the above calculation, taking into account the heat exchange effects of cold liquid entering the wick and therefore causing a change in the temperature profile.

A BRIEF EXAMPLE: A SEGMENT OF A PRIMARY WICK

Figure 3 presents a sample detailed model of a portion of a primary wick using a CAPPMP macro for vaporization, a SINDA node for the evaporator wall, a USER FTIE for the back conduction, and a WICK iface for the liquid-vapor interface motion. Instead of using the back conduction ftie, it is also possible to model the liquid side of the wick explic-

---

* Despite the name, the most common use of a FLAT iface is to subdivide a quasi-stagnant control volume with an imaginary film. Other types of ifaces are used to model bubbles, springs and bellows accumulators, pistons, etc.

† There are several ways to account for back conduction using ties and conductors, so long as it is not neglected.

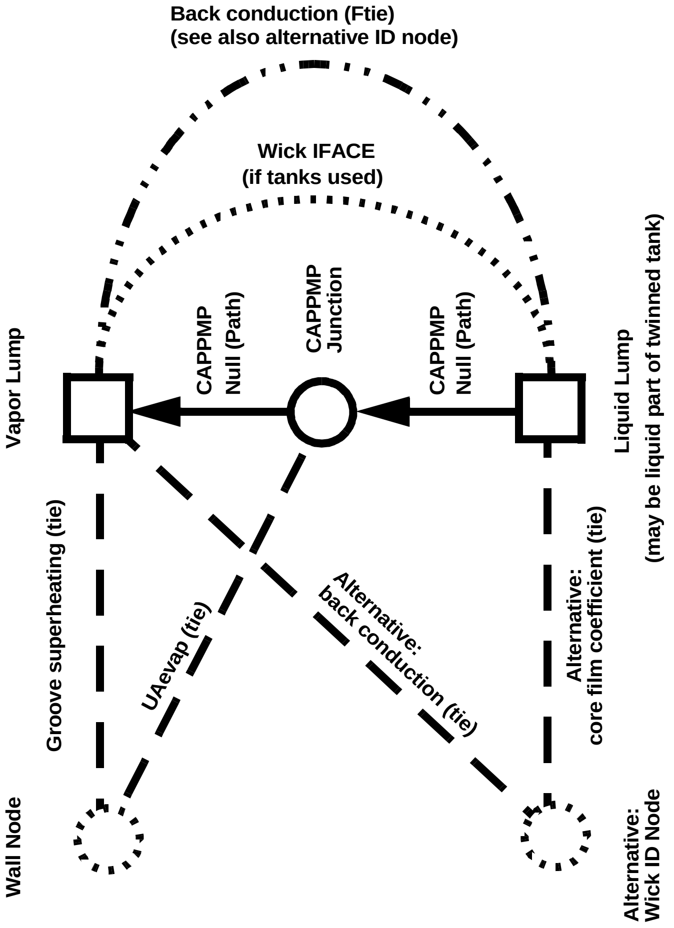


Figure 3: Two Possible Models of a Section of Primary Wick

itly using a SINDA node, as also shown in the diagram. Furthermore, the vapor side of the wick (usually equal to the saturation temperature) could be explicitly modeled separately from the evaporator wall node, but this alternative is not depicted.

The above discussion should reinforce one of the main theses of this paper: that SINDA/FLUINT network elements are building blocks only, intended to be used to create customized models of complex components. Few if any assumptions are made about which details are important for a specific LHP design, much less for each analytical case required to evaluate such a design. This approach provides tremendous flexibility and does not need to be replaced or augmented when a novel design is considered or when a change in the technology occurs. However, this generalized approach does place the burden on the user to ponder the physics being modeled and to understand the tools available to model such physics.

CHARGE TRACKING AND PRESSURE SELF-DETERMINATION

The internal pressure of a variable conductance CPL is determined solely by the temperature control of its reservoir. Similarly, if the temperature (and therefore pressure) of the compensation chamber of an LHP is fixed (e.g., by applying a heater and perhaps cooler), then the chamber will fill or empty as needed similar to a CPL.

However, LHPs are rarely used in this regulated mode, or if they are regulated, then the control point is not the compensation chamber itself but a more remote payload temperature. In these cases, the internal pressure of an LHP is like that of a fixed conductance heat pipe: it varies passively as needed to balance energy flowing in with energy flowing out.

In other words, for a given power input and sink temperature, there is a unique[*] temperature at which the compensation chamber will operate. A SINDA/FLUINT model must determine this operating pressure.

To understand why this can be problematic requires a more complete understanding of the difference between *tanks* (finite control volumes) and *junctions* (essentially, a tank in the limit of zero volume: an instantaneous tank). A tank's pressure is determined by the conservation of mass and energy: add more mass and it will usually rise in pressure, for example. Conversely, a junction's pressure is really defined as a change or delta from a neighboring lump's pressure according to the losses or gains in adjacent paths. Ultimately, a junction's pressure is therefore defined relative to tanks or plena (boundary conditions) in the loop. Therefore, a model cannot consist entirely of junctions: there is no reference state in such a network.

This is not a problem for a transient model using one or more tanks. If tanks are used almost everywhere, as is appropriate for a start-up model, then the pressures in the LHP will be self-determining. However, time steps can be small in such high-fidelity models: on the order of 0.1 seconds (0.001 to 1.0). To permit large time steps for thermally (and not hydrodynamically) dominated transients, junctions are normally used, and this may create a problem with the lack of a reference state.

Over the years, several solutions to this problem have been found for creating fast-executing thermal-oriented models with self-determining operating points. These include:

- Use one large tank to represent the compensation chamber, with junctions employed elsewhere. This tank can be larger than the actual volume of the compensation chamber and core: as a minimum it should represent the volume and mass of the whole loop. (This is currently the preferred method, and can employ variations such as the use of twinned tanks and/or the addition of a second tank representing the

---

[*] Neglecting hysteresis effects, and assuming the tilt and environment are invariant. Note that if the source temperature is fixed instead of the input power, then there may be two possible solutions: a high-conductance mode and a low-conductance mode, per the characteristic U-shaped LHP performance curves of source temperature versus power throughput.

high pressure vapor side of the loop, combined with a WICK iface in series.)
- Use a plenum for the compensation chamber, and use the Solver goal-seeking module to find the plenum pressure that balances energy in the loop. (This method is applicable only to steady state models.)
- Use a plenum for the compensation chamber, and use logic to dynamically adjust the plenum pressure such that energy flows in the loop are balanced. (This method is similar to that applied in vapor compression cycles.)

THE IMPORTANCE OF ABSTRACTION

One common pitfall of using SINDA/FLUINT is that it is *too* powerful: it provides the ability to model very complex physics (e.g., nonequilibrium two-phase heat and mass transfer with dissolution). When engineers lack the ability to include some physical phenomenon, they often dismiss it as negligible perhaps adding margins or conservatisms to compensate. However, the opposite is also true: when provided with the ability to avoid making such an assumption, engineers are tempted to include the more detailed physics just in case it matters.

Also, being visual beings, most engineers' attempts to model complex hardware such as LHP evaporators and compensation chambers are frustrated by excessive fidelity to the design geometry. Simplifying abstractions often result in much more efficient models that answer the required questions quickly.

Fast executing simplified models are often more valuable than slow executing high-fidelity models. They can be used to explore design sensitivities or uncertainties using parametric analyses or statistical design methods (using the Reliability Engineering module), or to size or select components (using the Solver optimization module), or to automatically correlate uncertainties to test data (using the Solver correlation module).

The ability to make intelligent modeling decisions and to avoid asking the wrong questions (and thereby getting side-tracked by unnecessary detail), requires a knowledge of both LHPs and SINDA/FLUINT.

Fortunately, most of these difficulties are associated with creating detailed transient models. If the user makes a few simplifying assumptions:

- single lump for the compensation chamber and wick core (neglecting secondary wicks, bayonets, orientation effects, etc.)

- pseudo-steady hydrodynamic response (i.e., steady-state or thermal effects only during transients)

then modeling LHPs becomes quite easy. In fact, it is only difficult to the extent that the condenser and its environment may require adequate spatial resolution (number of nodes and lumps). Developing such condenser/radiator models requires little expertise in either FLUINT or LHPs. A thermal engineer versed in SINDA or ESATAN but not in FLUINT nor LHPs would have little trouble developing such a model, especially if they use the "LHP Prebuilt" (described next) as a head start.

STARTING POINTS

A simple example of a steady-state LHP model is freely available (www.crtech.com) as an LHP *prebuilt*: a Sinaps*Plus* template from which a custom model can be made. Notes are included for how to convert the model into one suitable for analyzing thermal transients.

Models of fully hydrodynamic LHPs are used for analyzing start-up transients (Ref 3, 4) and other transients that affect secondary wick sizing. Unfortunately, no such fully transient prebuilts are available as starting points or examples since many variations are possible. However, extensive expertise is exists, and attempts are made to capture this knowledge and experience in the form of training notes that are freely distributed (www.crtech.com).

CONCLUSIONS

A comprehensive design and analysis environment exists that has features specifically targeted toward the challenging modeling requirements of LHPs and other capillary devices. This tool has been used for many years to produce, evaluate, and correlate the designs of LHPs and their kindred technology, CPLs.

This software is capable of simple steady-state sizing analysis as well as complex start-up transients including the effects of noncondensible gases. It is suitable for vehicle-level integration analyses. It is not hardware- nor application-specific, and so cannot be outdated by changes in technology or mission. Unlike proprietary codes, it is available to all LHP developers and to all of their customers, and models can be exchanged between all such organizations. A complete infrastructure of support and training exists, as well as many user interface programs and interconnections to structural and orbital analyzers, CAD drawings, etc.

However, this software requires a knowledge of LHP operation and of the thermophysical processes involved in those devices, at least for detailed studies. Often, there is a lack of agreement within the LHP community over the specifics of these processes.

In addition, potential users of LHPs are resistant to adopt this technology because of their inability to analytically integrate LHPs into their designs, and to confidently understand the results of tests.

It is ironic that both of these problems can be addressed by off-the-shelf analysis tools.

CONTACT

C&R Technologies, Inc.
303 971 0292
303 971 0035 (FAX)
www.crtech.com

REFERENCES

User's manuals, tutorials, and training notes are freely available in PDF format at www.crtech.com.

1. Cullimore, B. et al; "Thermohydraulic Solutions for Thermal Control, Propulsion, Fire Suppression, and Environmental Control Systems;" SAE 1999-01-2159.
2. Cullimore, B. et al; AIAA iface paper
3. Baumann, J. et al; "Noncondensible Gas, Mass, and Adverse Tilt Effects on the Start-up of Loop Heat Pipes;" SAE 1999-01-2048.
4. Jane's AIAA LHP paper