AIAA 2000-2503

Control Volume Interfaces: A Unique Tool for a Generalized Fluid Network Modeler

Brent A. Cullimore
David A. Johnson
C&R Technologies, Inc.

ABSTRACT

Over the past 15 years, the industry standard tool for thermal analysis, SINDA, has been expanded to include advanced thermodynamic and hydrodynamic solutions ("FLUINT"). With the recent culmination of the unique modeling tools, SINDA/FLUINT has arguably become the most complete general-purpose thermohydraulic network analyzer that is available.

Traditional network elements for fluid circuit analyzers include control volumes and flow passages. During the development of modeling tools capable of handling phasic nonequilibrium within SINDA/FLUINT, several new network elements were created as by-products. This paper describes one of them: control volume interfaces or "*ifaces*" for short.

Ifaces are used to describe how one control volume abuts another. While originally developed to model liquid-vapor interfaces within two-phase control volumes, they can also be used to describe pistons, spring bellows, liquid slugs, and curved interfaces such as those between bubbles and liquid as well as those within capillary structures (e.g., sintered wicks). More importantly, they can be used as an imaginary film to subdivide quasi-stagnant control volumes, extending the reach of a 1D network into certain 2D and 3D problems.

Despite their abstract nature, ifaces have been well received by analysts for a variety of modeling tasks.

INTRODUCTION

THERMAL/FLUID NETWORKS

SINDA/FLUINT[1] is a network-style analyzer. This means that while it solves large complex sets of simultaneous differential equations, the user does not *directly* define the equations to be solved. Rather, these equation sets are *indirectly* built by the user by instead specifying networks (circuits) of generalized modeling elements, which in turn generate the equations to be solved. These equations vary during the course of the solution (e.g., steady state vs. transient formulations, single-phase vs. two-phase formulations, etc.), but the network does not. Thus, the network becomes a user interface concept: the user need not concern themselves with the math as much as thinking about which building block is needed where.

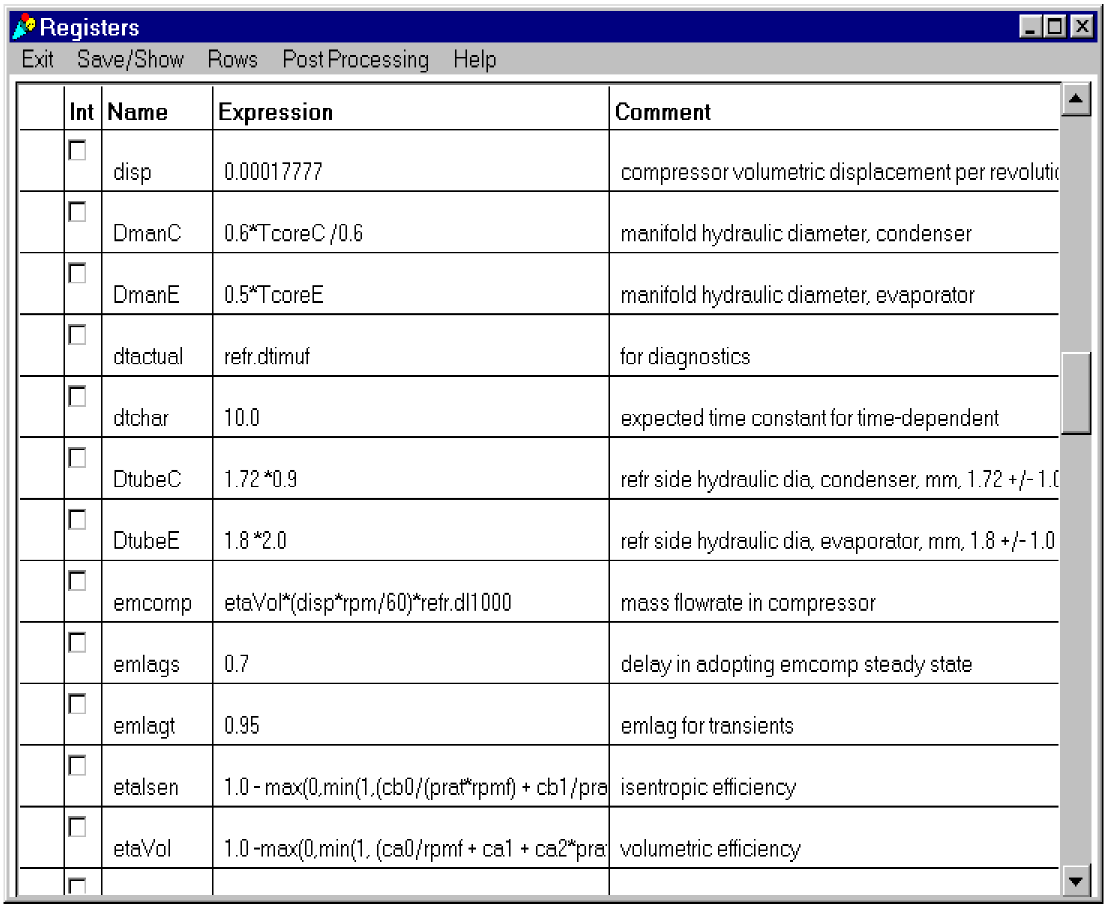
Complex hardware can be modeled using these generalized building blocks. This approach, plus an extremely flexible and extensible architecture, explains the long term success of SINDA/FLUINT in a wide variety of industries. Understanding this approach is also critical for understanding the innovations that ifaces represent.

OVERVIEW OF THE PROGRAM

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 electronics packaging, automotive, refrigeration, and power generation.

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 (Fig-



Registers

Exit Save/Show Rows Post Processing Help

Int	Name	Expression	Comment
☐	disp	0.00017777	compressor volumetric displacement per revoluti
☐	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 +/- 1.0
☐	DtubeE	1.8 *2.0	refr side hydraulic dia, evaporator, mm, 1.8 +/- 1.0
☐	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
☐	etaIsen	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

ure 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-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.

<u>FLUINT</u>

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.

Single- or two-phase flow can be modeled either for pure components (e.g., steam and water), for nonvolatile/noncondensible mixtures (e.g., air and oil), and for condensible/volatile mixtures (e.g., air and oil and steam and water). Gases can dissolve into or evolve from the liquid phases according to saturation relationships and finite rate mass transfer. Up to 26 nonreacting substances can be mixed within each fluid submodel, and up to 25 fluid submodels can be used.

Two-phase flow is by default homogeneous (uniform velocity: equal liquid and gas velocities) and in phasic equilibrium (perfectly mixed: equal temperatures and pressures between phases). However, it is a simple matter to elect the prediction of flow regimes, to model slip flow (unequal liquid and gas velocities), to model phasic nonequilibrium in quasi-stagnant volumes and within duct flows, and to model nonequilibrium expansions in valves, orifices, and venturis.

Table 1: SINDA/FLUINT Hierachy of Modeling Options

- **THERMAL/FLUINT MODELS**
  - **Registers, Expressions, 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- and time-varying
      - Radiation
        - Temperature- and time-varying
    - *Sources*
      - Temperature- and 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-calculated convection conductance
    - *Duct macros* (subdivided pipelines)
    - *Capillary evaporator-pumps* (CAPPMP macros)
    - *Ifaces* (control volume interfaces), w/ or w/o inertia
      - flat (zero pressure difference)
      - offset (finite pressure difference)
      - spring (i.e., bellows, etc.)
      - spherical bubble
      - wick (liquid-vapor within 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**

Unique features such as time- and direction-varying body forces and capillary device models are important to the aerospace industry. Because they are unique, such tools have found uses in nonaerospace applications such as modeling rotating machinery.

TANK VOLUME OPTIONS

"Tanks" are the fluidic control volumes within SINDA/FLUINT. Although the phase and species within a tank can change during the course of the solutions, by default the wall is rigid.

Instead of a fixed tank size, it is also possible to define the volume rate of change (VDOT = dV/dt) for any tank, and for this rate (like any other SINDA/FLUINT parameter) to be variable within a run, perhaps defined by an equation or subroutine. Although not commonly used, this dV/dt term has been applied to models of reciprocating pistons, the human heart, and scroll compressors.

More commonly, the rigid wall assumption is eliminated by supplying a *compliance*, defined as COMP = (dV/dP)/V, or the fractional change in volume (dV/V) per unit change in pressure (dP). This definition is useful because compliance then becomes a constant when representing liquid and/ or container wall compressibility (a common action for modeling waterhammer and other fast transient events), or when expediently representing gas inclusions without explicitly modeling the phase or constituent represented by the inclusion.

Compliance can be combined with the stretch or shrinkage rate (VDOT) to yield a dependent implicit differential equation:

$$V(t) = VDOT + V(t)*COMP*dP(t)/dt$$

This equation is "dependent" because volume is not normally part of the solution vector for the equation set. Rather, the solution vector for a tank containing a pure substance is given by either $(\Delta P,\Delta U)$ for liquid-filled tanks and $(\Delta M,\Delta U)$ for two-phase or vapor/gas tanks, where ΔP is the differential pressure $(P(t+\Delta t) - P(t))$, ΔM is the differential mass, and ΔU is the differential extrinsic internal energy. The above equation is folded into the two (mass and energy) equations for a tank, and then the resulting differential volume, ΔV, can be calculated from the resulting solution vector.

CONTROL VOLUME INTERFACES: OVERVIEW

Flow passages ("paths" in FLUINT) are common modeling elements in a variety of network-style fluid analyzers. They represent flow through pipes, orifices, pumps, filters, etc. In other words, they transport mass according to the pressure drop (along with other considerations) between an upstream point and a downstream point.

Ifaces are like flow passages in that they connect one control volume to another, but instead of transporting mass, they represent the movement of the boundary between control volumes (Figure 2).

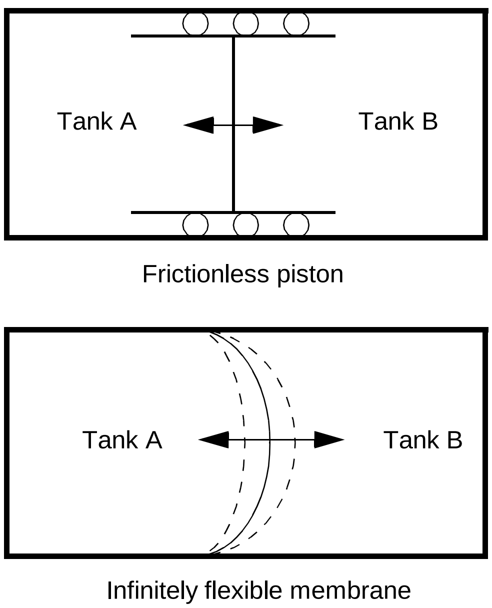


Figure 2: Control Volume Interface Examples

This interface may be a flexible membrane or diaphragm, a bellows, a piston, or a slug of liquid. As the name implies, it might also represent a liquid/vapor interface, where liquid is on one side of the iface and vapor is on the other. This liquid/vapor interface might be flat (no pressure difference between phases), slightly curved (as in a small bubble), or highly curved (as might occur within a capillary structure).

The interface may also be used as an imaginary boundary separating control volumes, and therefore permitting arbitrary subdivision of a quasi-stagnant control volume. This important interpretation of ifaces is frequently exploited in the modeling of thermally stratified cryogenic dewars, as will be described later.

Ifaces may be used in parallel with paths (flow passages) and fties (heat transfer within the fluid). However, ifaces can only be used between tanks, since only they have definite volumes. In certain steady state solutions, where tanks are treated like junctions, ifaces disappear from the equation set.

ASPECTS COMMON TO ALL IFACES

There are many types of ifaces, as will be described below, but they all share certain features.

First, they all have the option of adding an inertia term to encompass the case of boundaries with significant mass (i.e., a large piston, a slug of liquid, or the *added* or *virtual* mass of a liquid/vapor interface). This term is defined as the mass of the interface divided by the square of its area: M/A^2.

Second, the user can define upper and lower volumetric limits (VHI and VLO, respectively) on the range of action for an iface, as depicted in Figure 3 for the case of a bellows accumulator. When the volume limits are exceeded, the iface temporarily turns off. If the forces on the interface are such that the active regime is again in effect, the iface will reappear.

Third, ifaces use "duplication factors" to exploit symmetries in the system for reduced model sizes. For example one small bubble can be duplicated or magnified according to the number of actual bubbles being modeled. Further discussion of duplication factors, which are also applicable to paths, ties, and fties, is beyond the scope of this paper.

IFACE TYPES

This section describes the different types of iface elements available for modeling purposes.

FLAT

The FLAT iface simply keeps the pressures in adjacent tanks equal. It is named after a flat liquid-vapor interface, which exhibits no capillary forces. However, FLAT ifaces can be used in the absence of two-phase flow, and are in fact the most commonly applied iface.

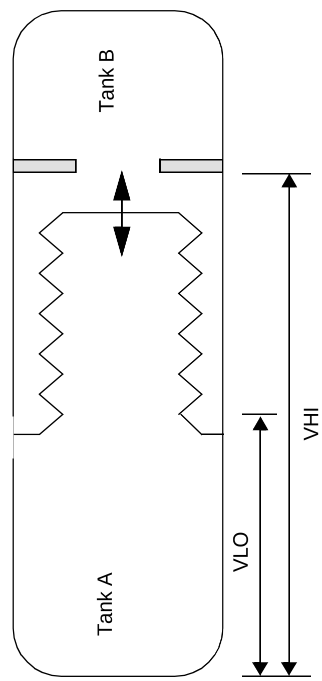


Figure 3: Volume Limits as Applied to Bellows Accumulator Example

For example, FLAT ifaces are used to vertically subdivide the ullage and/or liquid in thermally stratified cryogenic dewars. In this example, the pressures in the tanks may not be equal due to hydrostatic forces, and this body force term is permitted as a "violation" of the strict interpretation of a FLAT iface.

Other examples include a horizontal piston or liquid slug separating two control volumes, with or without mass. If the piston or slug is vertical, then the pressure in the lower tank will be higher than in the upper tank, and an OFFSET iface should be used instead (described next). The mass of an interface can cause both a force and a inertia. The inertial term for any other iface represents a lag in achieving the specified state (which is zero pressure difference, in the case of a FLAT iface).

OFFSET

The OFFSET iface is almost as simple as the FLAT iface, but allows the pressures in the adjacent tanks to vary by a fixed* amount. The OFFSET iface is intended to model vertical pistons or liquid slugs, but can be used as a general modeling tool combined with user-supplied logic.

SPRING

The SPRING iface obeys a spring-force relationship such as $F = k(x-x_o)$. Cast in the form of a pressure/volume relationship, the SPRING iface obeys the formula $\Delta P = k(V-V_o)$ where k is the spring constant (dP/dV).

SPRING ifaces are intended to help model bellows accumulators, stiff diaphragms, and other stretchable dividers.

SPHERE

The SPHERE iface can only be used in two-phase flows. It models a spherical bubble where the pressure difference is proportional to the bubble size, according to the capillary forces developed across the interface. In other words, liquid is on one side of the SPHERE iface, and vapor is on the other side.

Multiple bubbles can be modeled using multiple ifaces and control volumes, perhaps using the duplication factors to reduce the model size if many of the bubbles can be assumed to be same size.

It should be noted that bubbles are rarely modeled directly using SINDA/FLUINT: the SPHERE iface is therefore rarely used. Instead, bubbly flow can be modeled using correlations that average interphase friction and heat and mass transfer over a distribution of bubble sizes.

WICK

SINDA/FLUINT is frequently used for modeling capillary heat transport devices such as capillary pumped loops (CPLs) and loop heat pipes (LHPs), as described in Reference 2. It is also used for modeling capillary acquisition devices and fluid gaging devices in fuel tanks. Therefore, it has many modeling options unique to that specialized type of analysis. The WICK iface is one such capillary feature.

The WICK iface is somewhat like the SPHERE iface in that liquid is presumed to be on one side,

* Like any other parameter in SINDA/FLUINT, this offset need not truly remain fixed, but can vary according to user-specified equations or Fortran-style logic.

while vapor is on the other. It is also like the SPRING iface in that it obeys a nontrivial relationship between volume and pressure, as depicted in Figure 4. This relationship includes a flooded state somewhat like a FLAT iface, a subsaturated state somewhat like an OFFSET iface, and an intermediate state somewhat like a nonlinear SPRING iface.

WICK ifaces can be linked with other capillary paths in FLUINT to include the effects of wick pressure drop, vaporization at the surface, etc. Because WICK iface operation is complicated and is only used in very specialized application, no more details will be given here.

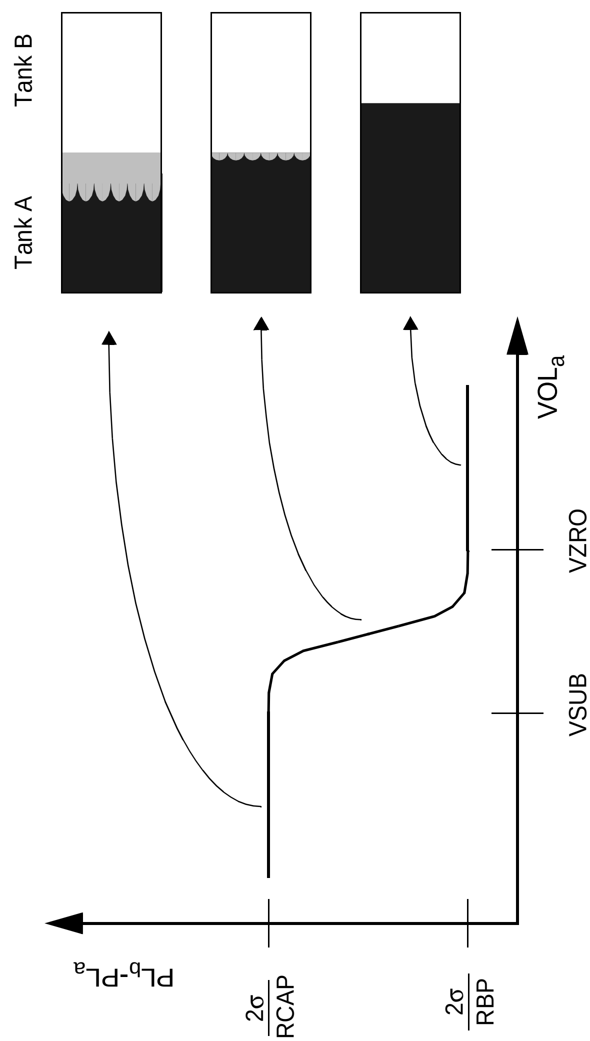


Figure 4: Steady Pressure-Volume Relationship for a Wick IFACE

NULL

The NULL iface represents direct access to the somewhat abstract terms that go into the matrix solution, and it is therefore applicable only for advanced users. FLUINT has many such layers of access depending on the skill and need of the analyst.

SUBVOLUMES: NETWORKS OF INTERFACES

When two tanks are joined by an active iface, they form a *subvolume*. The total volume of this subvolume must be conserved: if one tank grows, the other must shrink.

Compliances and VDOT terms applied to individual tanks are taken into account in this “conservation of volume” relationship, but they are applied to the entire subvolume. In other words, the total growth or shrinkage rate of a submodel is a function of the COMP and VDOT terms for the constituent tanks in a subvolume. For example, if one compliant tank is connected to another liquid-filled rigid-walled tank via an iface, the second tank will effectively respond to the compliance of the first tank because the boundary between them is itself flexible.

If one of the two tanks is connected to a third tank via an iface, then the three tanks form a subvolume. This can continue indefinitely. If there are N tanks in a fluid network, there can exist anywhere from 0 to N/2 subvolumes in the network at any time. Because subvolumes can join and subdivide spontaneously as ifaces turn on and off (perhaps from hitting a volume limit, or perhaps from being turned off by the user), FLUINT must continuously track these subvolumes and reorganize the volume conservation equations for each (see “Solution Methods” below).

More than one iface cannot be used in parallel between two tanks, since to do so would represent either overspecification or unnecessary redundancy. Similarly, the subvolume formed by the three tanks and ifaces depicted in the top half of Figure 5 overprescribe the problem.

FLUINT automatically detects and disconnects any such redundant ifaces (bottom half of Figure 5), issuing warnings or stopping only if it is unable to unambiguously resolve these overspecifications.

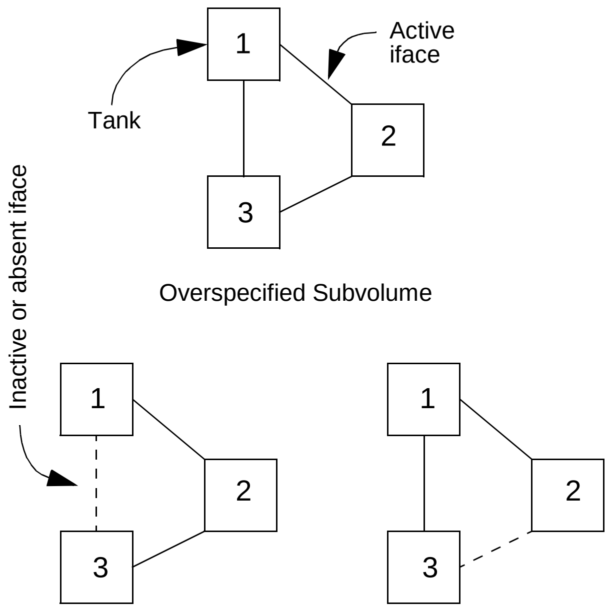


Figure 5: Overspecification in Submodels

EXAMPLE APPLICATION

To illustrate the non intuitive behavior of ifaces, consider a stack of gas-filled tanks interconnected by tanks representing an axial subdivision of a long, thin pressurized bottle. The tanks are interconnected by paths, perhaps duct elements.

If the tank is depressurized by releasing mass from one end, then all tanks will drop in pressure and will decrease in mass, but their volumes will remain constant. Axial flow through the bottle will cause a very slight pressure gradient.

Now consider replacing the paths with FLAT ifaces and repeating the analysis, as shown in Figure 6. Each tank drops in pressure as before, but instead of their mass decreasing, their volume increases instead.

Only the end tank (#10) loses mass to the exhaust. It therefore shrinks until it is so small that (1) the analysis must be terminated, (2) the iface between tanks #9 and #10 needs to be shut off (by specifying a lower volume limit) and replaced by a flow path, or (3) the exhaust must be moved to the next tank (#9) to continue depressurizing.

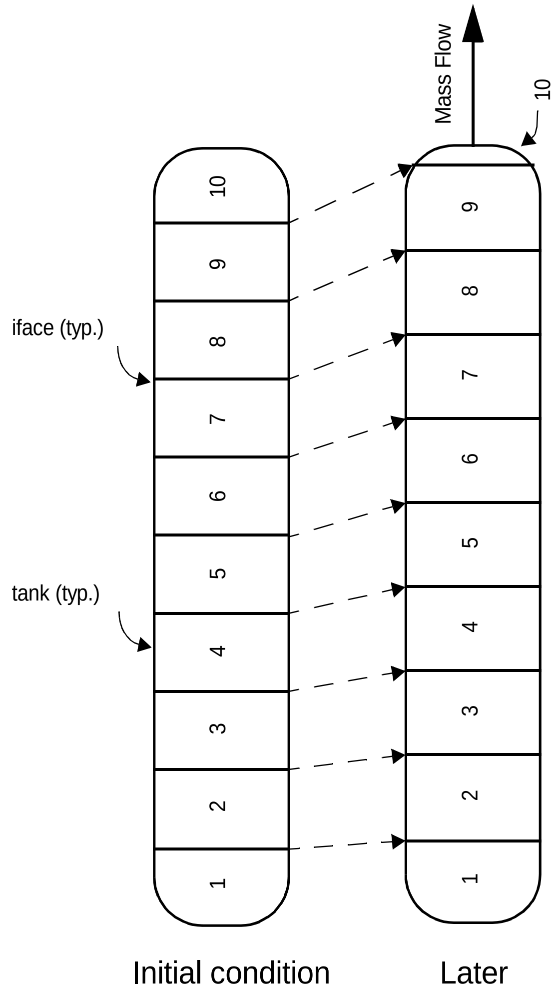


Figure 6: Axial Subdivision of a Depressurizing Vessel

SOLUTION METHODS

When the volumes of individual (non interfaced) tanks change, this volume change is predicted by inserting terms into the mass and energy equations as described above.

When interfaces exist, an extra solution term ΔV exists for each tank within a subvolume: ($\Delta P, \Delta U, \Delta V$) or ($\Delta M, \Delta U, \Delta V$). In other words, the volume equations become independent of the mass and energy equations when ifaces are applied.

If there are M tanks within a particular subvolume, this generates M unknowns: the differential volume ΔV for each tank within that subvolume. This is matched by M equations: M-1 pressure/volume relationships for each iface in the subvolume, plus a final "conservation of volume" equation for all tanks in the subvolume.

CONCLUSIONS

A powerful network element, the control volume interface, has been introduced to a thermal/fluid circuit analyzer. This element is apparently unique: the authors are unaware of any analogous building block in any other tool.

Ifaces were originally developed for use with the phasic nonequilibrium features in SINDA/FLUINT, in which a single two-phase tank can be subdivided into *twinned* tanks, one of which models the bulk liquid while the other models the bulk vapor. SINDA/FLUINT automatically places an iface between such paired tanks.

Once created for this purpose, they were generalized and made available as a general-purpose building block element in a SINDA/FLUINT fluid network.

Despite its abstract nature, the iface has been readily accepted by the analysis community in part because it solves long-standing problems modeling thermally stratified tanks and the dynamic motions of liquid-vapor interfaces within capillary wicks.

ACKNOWLEDGMENTS

SINDA/FLUINT would not exist were it not for the continuing support of the Crew and Thermal Systems Division of the NASA Johnson Space Center. Dr. Eugene Ungar and Cynthia Cross were the technical advisors for the projects that resulted in ifaces, among other improvements.

CONTACT

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

REFERENCES

User's manuals, tutorials, and training notes for all software discussed 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; "Steady-State and Transient Loop Heat Pipe Modeling," SAE 2000-TBD, ICES Conference, Toulouse, France.