SAE 03ICES-021

Adding Heat Pipes and Coolant Loop Models to Finite Element and/or Finite difference Thermal/Structural Models

B. Cullimore, D. A. Johnson
C&R Technologies, Inc.

ABSTRACT

Active cooling technologies such as heat pipes, loop heat pipes (LHPs), thermosyphons, loop thermosyphons (LTSs), and pumped single- or two-phase coolant loops require specialized modeling treatment. However, these 1D ducted systems are largely overlooked in 3D thermal modeling tools. The increasing popularity of CFD and FEM models and generation of analysis data from 3D CAD data are strong trends in the thermal analysis community, but most software answering such demands has not provided linear modeling elements appropriate for the simulation of heat pipes and coolant loops.

This paper describes techniques whereby CAD line-drawing methods can be used to quickly generate 1D fluid models of heat pipes and coolant loops within a 3D thermal model. These arcs and lines can be attached intimately or via linear contact or saddle resistances to plates and other surfaces, whether those surfaces are modeled using thermal finite difference methods (FDM), or finite element methods (FEM), or combinations of both. The fluid lines can also be manifolded and customized as needed to represent complex heat exchangers and plumbing arrangements. Furthermore, the assumption of 1D flow can be combined with 2D/3D models of walls, including advanced methods of extruding a complex 2D cross-section along a curved or mitered centerline.

To demonstrate these concepts, several distinct examples are developed and discussed.

PRIOR STATE-OF-THE-ART

3D THERMAL MODELING

A variety of network-style thermal conduction/capacitance modeling tools exist, including Thermal Solution's Sauna®, Network Analysis' SINDA/G®, Thermal Associates' TAK, and the SINDA side of C&R's SINDA/FLUINT. Usually these codes are erroneously considered “finite difference” when in fact they are geometry-independent thermal network (circuit) solution engines that can be used to solve not only finite difference problems and 1D lumped parameter problems, but also finite element problems (with proper input preparation). They usually feature concurrently executed user logic and/or other equation-style inputs. Increasingly, thermal network analyzers are used with graphical user interfaces (usually geometry-based) that help prepare inputs, although most can still be accessed at the “thermal circuit level.” Such access is important for high-level lumped parameter modeling in which a complex component such as a battery might be represented using effective thermal mass, conduction, surface area, etc., or where incorporation of compact models is required.

Similarly, there is no shortage of software tools for modeling steady or transient conduction within shells or solids, usually using finite elements (e.g., MSC/NASTRAN®), occasionally using finite differences (e.g., SDRC's TMG®), and in at least one instance (Ref 1) both finite elements and finite differences can be used in a mix-and-match fashion. Indeed, almost every finite element method (FEM) structural analysis program offers such “heat transfer modeling” as an option. Most of these analysis codes also supply means of generating models from CAD data, albeit with varying degrees of flexibility.

At the very least, structural FEM models can be generated from CAD representations using a wide variety of software. Unfortunately such models, being based on structural meshes, are rarely appropriate for direct use as thermal models. Few of the available surface and solid (2D/3D) codes are specifically designed for thermal management tasks. Only those that are so oriented tend to support analysis of higher level assemblies critical to product-level heat transfer, including effects such as contact conductance and efficient radiation calculations. Few provide any fluid flow capabilities, excepting those that use full CFD (e.g., Fluent's IcePak®, SDRC's ESC®).

A few other 2D/3D codes provide fluid flow networks. With one exception (Ref 2), most of this class of software require that *answers* such as flow rates and heat transfer coefficients must be supplied as *inputs*. Worse, interconnections with the 2D/3D thermal geometry are not automated. Alternative graphical user interfaces for flow network solvers are based on schematics with the surfaces and solids associated with the thermal model either absent or oversimplified (e.g., Ref 3): the emphasis is either placed on the 1D fluid modeling, or on the 3D thermal modeling, but not on both in the same package.

In summary, most thermal engineers have access to or can relatively easily generate 2D/3D thermal conduction models, and some can generate models with thermal features such as contact conductance and radiation, but few have been able to link air flow or ducted coolant flow modeling into these models without resorting to a full 3D CFD solution.

CFD MODELING

Ducted single-phase flow with heat transfer may be modeled using a variety of 2D/3D CFD methods. However, very small CFD elements or volumes are required within the boundary layers of objects in freestream (unducted) flow, and computational resource requirements usually increase geometrically with increased discretization. In adiabatic ducted flow, CFD elements must be small throughout the model. In ducted flow with heat transfer, most CFD codes require even smaller elements to avoid large error terms in estimating conjugate heat transfer at the wall.

The cost of solving these models is very high for realistically complex systems such as an entire coolant loop, thereby making transient analyses essentially untenable. Even making parametric or iterative steady-state runs can be too time consuming, especially since few CFD codes offer full parametric modeling capabilities: model and mesh changes are difficult to make *between* runs much less *within* runs.

Two-phase flow with phase change, such as occurs in heat pipes (including loop heat pipes), thermosyphons, spray coolers, and vapor compression cycles is currently beyond the realm of practical commercial CFD modeling for system-level modeling.

For these reasons, some CFD providers have recently begun to offer 1D flow modeling alternatives, recognizing that the above limitations are likely to remain intractable for many years to come.

1D FLOW MODELING

One dimensional flow models might still be called “computational fluid dynamics” by some engineers, but 1D models are distinguished by the complete elimination of the mesh in the nonaxial dimensions. Instead, well-established empirical correlations are used for both heat transfer and pressure drop. In other words, the boundary layers in 1D duct flow are not solved from “first principles” as in a CFD approach, but rather using computationally efficient assumptions based on copious testing. Because the radial and circumferential dimensions do not need to be discretized, even the axial dimension does not usually require as much subdivision as it would in a CFD approach. Thus, 1D flow models are many orders of magnitude faster to solve than are 3D flow model for ducted systems.

In the 1D approach, momentum conservation is applied axially, with wall friction applied to the axial flow momentum equation using correlations appropriate for the duct shape, fluid, current flow rate, etc. In other words, the only “velocity field” is a single vector in the axial direction (at any point along the flow stream).

Energy and mass (and species etc.) can be conserved at axial points along the flow direction. Heat transfer coefficients can vary around the circumference in a quasi-2D fashion, again using an empirical approach. There is no subdivision of the fluid control volumes in the radial or circumferential directions, resulting in simple fast-solving network schematics.

For single-phase flow, the speed enhancements over CFD methods are dramatic. For two-phase flow, the 1D approach is “enabling” since such problems are essentially intractable using 3D CFD approaches, which must resolve and track each phase and must handle both the sharp gradients and the intense coupling with thermodynamics and heat transfer that is required in two-phase flows.

A “first principals” CFD approach (i.e., eliminating Reynolds- and Nusselt-based correlations) is considered by some engineers to be more accurate. While this opinion is difficult to defend for ducted flows, there *are* some circumstances where an empirical 1D approach is strained. One example is two-phase flow, where 20% error in predicted friction or heat transfer coefficients would be considered “excellent” in the empirical correlations underlying a 1D flow model. Fortunately, the fast solution speed of 1D methods enables higher-level methods for dealing with such uncertainties (Ref 4, 5).

1D solution speeds also allow detailed transient analyses to be made, along with rapid model changes (including parametric sweeps during a single solution run). Such parametric model changes are important precursors for higher-level analyses and design activities such as automated sizing, selection, and location of components (Ref 6).

In summary, the “loss” of the extra mesh dimensions yields an enormous gain in solution speed, and this gain can be applied to higher-level engineering tasks rather than to single “point design simulation” (i.e, predicting how a single design point responds to a single scenario). 1D flow solutions are clearly superior to 2D/3D CFD solutions for ducted flow problems.

However, one problem has existed with the 1D flow network modeling approach for thermal modeling: the lack of integration with 3D thermal models.

1D FLOW MODELING WITHIN 3D THERMAL MODELS

Reference 2 introduced a methodology for building 1D ducted or unducted flow models within 3D (i.e., FDM and/or FEM) thermal models using FloCAD®, a module of C&R’s CAD-based Thermal Desktop®. Selecting 1D methods for unducted flow requires that simplifying assumptions be made. While such simplifications are not always appropriate for modeling unducted air flows, they *are* appropriate for ducted air or coolant flows, as was discussed above.

However, significant expansions of the methods detailed in that reference were required in order to apply them to ducted flow systems such as coolant loops, heat pipes, and refrigeration systems. Specifically:

- Means had to be supplied of drawing free-form lines and arcs using CAD tools, and then enabling these 1D lines elements to be considered as either pipes or ducts (for coolant loops, loop heat pipes, loop thermosyphons, vapor compression cycles, etc.) or as constant or variable conductance heat pipes (CCHPs, VCHPs).
- These fluid lines, whether representing ducts or heat pipes, had to be able to optionally include the pipe wall or container without violating the 1D assumption: 1D thermal conductive/capacitance network elements were required.
- In cases where 1D thermal models of the pipe or container were inappropriate, new methods for extruding cross-sections along centerlines had to be developed.
- The fluid lines had to be attachable to thermal solids and surfaces with appropriate models for fins, saddles, bonds, contact conductance, etc.
- The fluid lines had to have variable axial resolution, and yet be able to be subdivided as needed to form tees, manifolds, etc.
- The axial discretization of the fluid lines needed to be specifiable independent of the spatial discretization and modeling method (FEM vs. FDM) of the surface or solid to which the fluid line was to be attached.

These improvements have been completed successfully, yielding a methodology uniquely suited to cooling applications requiring ducted air, heat pipes, or single- and two-phase coolant flow networks.

Several diverse examples will be presented, each detailing compliance with the above modeling requirements.

HEAT PIPE WITH 1D WALL

System-level models of heat pipes can be relatively easily modeled using network methods, such as those outlined in Reference 7. In these methods, the vapor state is represented by a massless node, and all active (i.e., evaporating and condensing) wall segments area connected to this central node via conductances or resistances (Figure 1).

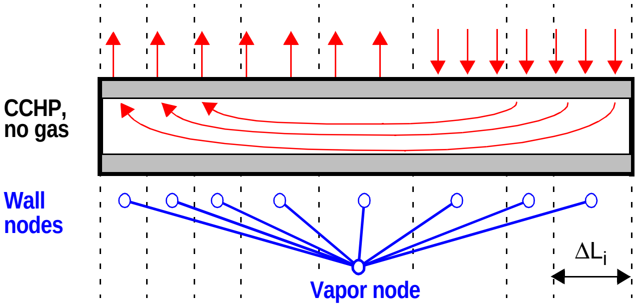


Figure 1: Simplified 1D Network Model of a Heat Pipe

Such methods are intended for users (not developers) of heat pipes, requiring only readily available data (Figure 2) and providing performance metrics such as the power-length product (QL_{eff}), which is used to determine how close the heat pipe is operating to vendor-supplied limits. These methods can be readily extended to include blockage by noncondensible gases (NCGs), whether intentional (VCHPs) or accidental (CCHPs).

While more complicated representations are discussed later, the simplest representation for the pipe wall itself is a 1D finite difference line. The axial resolution of this line can be adjusted as needed, and limited radial resolution (e.g., nodes at the inner and outer diameters) is possible, but circumferential gradients are neglected.

With this assumption, the heat pipe can be represented by a simple line, or by a complex line composed of straight segments, arcs, etc. such as that depicted in Figure 3. In that example, a CCHP is depicted as three straight sections joined by two 90 degree bends. The straight sections happen to coincide with the condenser, evaporator, and transport section, but in the more general case there is no need to formally designate such zones: hot portions evaporate, cold portions condense, and that condition can change at any time as the solution proceeds.

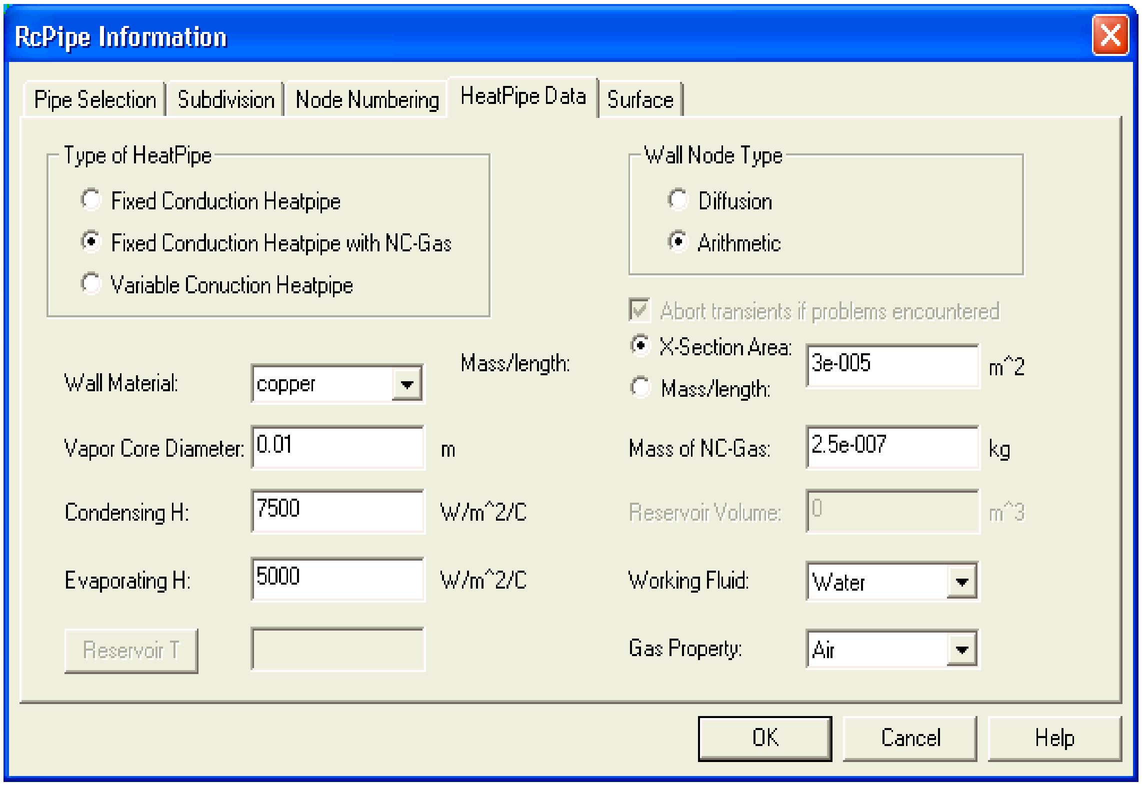
The pipe is attached via lineal or areal conductances to finite difference surfaces (finite element examples will be provided later) representing a bond or contact. In the leftmost case in Figure 3 with no NCG, the condenser of the pipe is isothermal. As NCG is added (simulating degradation or aging mechanisms), the far end of the condenser becomes blocked and a temperature gradient appears in the lower plate.

In Figure 3, the pipe itself is barely visible. In fact, its size has been augmented in those screen captures by making modeling elements (e.g., nodes) visible along its length. Many analysts today expect geometric fidelity: a pipe should look like a pipe and not a line, despite the fact that the underlying model is 1D and a 1D solution is appropriate. For these users, and more importantly for cases where 1D wall models *are* insufficient, 2D examples will be provided later.

FLUID LOOPS WITH 1D WALLS

Loop heat pipes (LHPs) are increasingly popular devices for managing heat transport, especially across flexible joints (Ref 8). While the simple models used for heat pipes are completely inappropriate for LHPs models, validated methods exist using thermohydraulic codes (Ref 9, 10, 11).

Figure 4 shows an LHP used as a replacement for the transport and condenser sections of the heat pipe in the previous example. Notice that the serpentine condenser of the LHP is able to attach to more of the rejection plate, such that the extra interface (heat pipe to LHP) and constriction resistances of the small evaporator are more than compensated: the total thermal resistance is less because spreading resistances in the sink plate have been lowered.



Type of HeatPipe

○ Fixed Conduction Heatpipe

● Fixed Conduction Heatpipe with NC-Gas

○ Variable Conuction Heatpipe

Wall Node Type

○ Diffusion

● Arithmetic

☑ Abort transients if problems encountered

Wall Material: copper

Mass/length:

● X-Section Area: 3e-005 m^2

○ Mass/length:

Vapor Core Diameter: 0.01 m

Mass of NC-Gas: 2.5e-007 kg

Condensing H: 7500 W/m^2/C

Reservoir Volume: 0 m^3

Evaporating H: 5000 W/m^2/C

Working Fluid: Water

Gas Property: Air

Figure 2: Input Form for CCHPs and VCHPs

The serpentine condenser is yet another 1D element: this time a simple pipe wall perhaps chosen from a piping schedule. Its path through three dimensional space is more complicated, with out-of-plane arcs and lines. Once again, the line is barely visible by itself, with modeling elements (in this case, FLUINT paths depicted as arrows) highlighted for increased visibility. (Visibility is a concern for screen captures, but not for code usage.)

Figure 5 depicts another case employing a 1D pipe wall, but it illustrates two important distinctions. First, the pipe consists of manifolded segments. Unlike heat pipes, such fluid pipes can be joined at tees. Pipes with coincident points can be optionally linked to enable both fluid and energy (through a conductive wall, if present) to be transferred through the resulting joint.

The second distinction is that the plate to which the pipe is attached is built using finite elements instead of finite differences. This choice enables smoother thermal contours during postprocessing, but doesn't otherwise contribute to increased accuracy compared to finite difference approach for basic shapes. For more complex shapes, finite element methods are preferred.

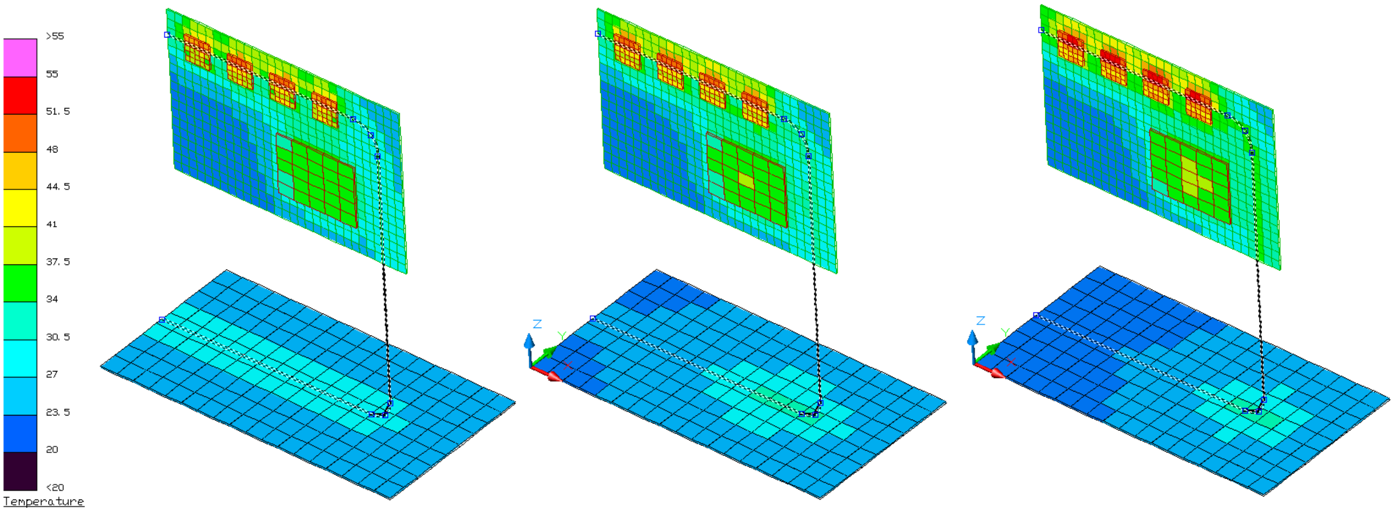


Figure 3: Constant Conductance Heat Pipe with Increasing Noncondensible Gas

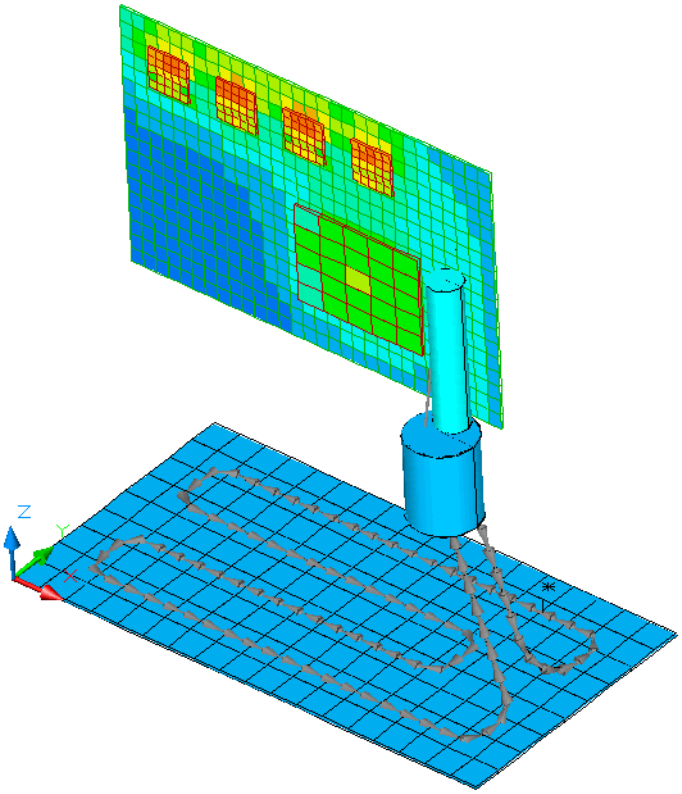


Figure 4: LHP with Serpentine Condenser on FDM Plates

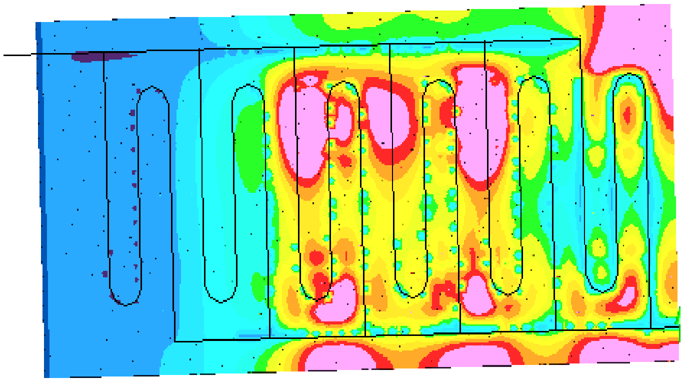


Figure 5: Manifolded Coolant Loop on FEM Plate

Capillary pumped loops (CPLs) are a kindred technology to LHPs. Figure 6 depicts another serpentine condenser, one used on a space shuttle experiment in the mid 1980s. As with the previous example, the underlying plate is modeled using finite elements. Unlike the previous example, however, a 3D model of the plate was chosen because of its significant thickness. The pipes themselves, however, remain 1D elements despite their apparent thickness: they are depicted as "ribbons" in the CAD drawing with the thickness assigned to be the same as the actual pipe diameter for visual clarity. However, since the thickness of the *model* (vs. *depiction*) is zero with respect to contact calculations, the pipe is intrinsically assumed to be small compared to the nodal resolution of the plate.

2D WALLS

The 1D wall assumption is not always possible. For example, the pipe or heat pipe might be large compared to the nodalization of the attached surface or solid, such that the pipe wall contacts several nodes across its width. Also, the pipe might not be attached along a side, but might instead be placed in a milled channel or might intersect perpendicular cooling fins. In such cases, a 2D wall model might be needed even if the heat pipe or flow model remains 1D.

Another reason to switch from a 1D to a 2D wall model is if a temperature gradient exists circumferentially. For example, heat pipes are commonly bonded within honeycomb panels and communicate thermally with each facesheet. In such a case, the heat pipe can evaporate on one side of its cross-section and condense on the other side.

Finally, a 1D model cannot participate in radiation heat transfer, where the freedom to abstract a model from its graphical depiction is necessarily restricted owing to the geometric nature of such calculations. Similarly, the existence of a definite 2D (vs. virtual 1D) wall geometry assists in calculating convection and other heat gains and losses off the wall itself.

For these reasons, and for those analysts who want more fidelity between the *model* and the *depiction* (usually in order to produce more impressive presentation materials), the above 1D modeling methods have been recently expanded to include 2D wall models.

Figure 7 depicts an example of a 2D heat pipe (Ref 12). In this case, the heat pipe is represented by a simple rectilinear cylinder despite the fact that its size is small compared to the resolution of the source plate. The need for a 2D model is driven instead by the rejection end of the heat pipes. The pipe wall intersects the heat sink fins perpendicularly, and those fins are too small to permit the pipe "contact" to be represented by a single node: a circular intersection is required instead.

Similarly, the air-cooled vapor compression cycle condenser depicted in Figure 8 demands a 2D pipe because of another perpendicular intersection and the large size of the pipe relative to the fins. In this instance, the pipe wall is modeled with finite differences while the fins (whose shape is irregular due to the cut-out for the pipe) is more appropriately modeled using finite elements.

In many applications for 2D pipe and heat pipe models, the centerline of the pipe is not a straight line, nor is the cross-section a simple cylinder. Instead, the centerline might be a complex curve in three-space (say, for example, the centerline of the LHP condenser in Figure 4). Also, the cross-section might be arbitrarily complicated.

The challenges for such modeling are significant. The cross-section must be extruded intelligently through miters and arcs, such that complex arrangements can be quickly built requiring minimal user specifications and guidance.

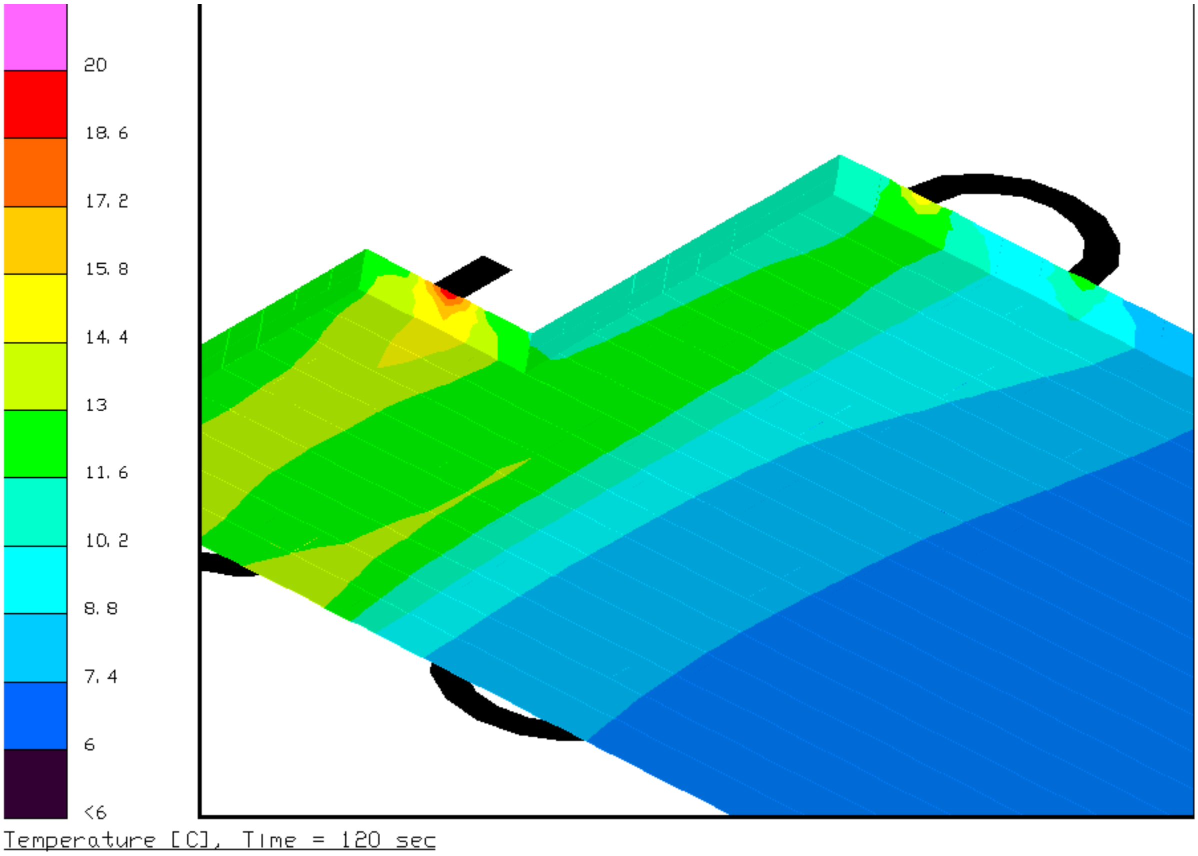


Figure 6: CPL Condenser with 1D Wall on 3D FEM Solid

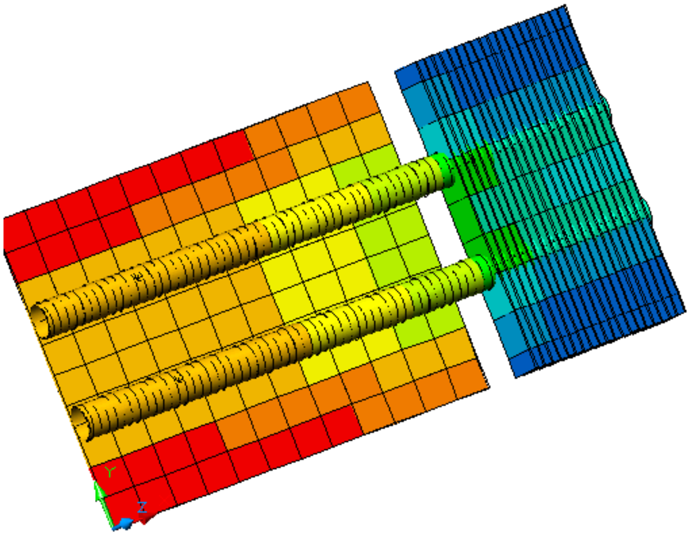


Figure 7: 2D Chip-to-fin Heat Pipes

Also, the 2D walls must respect the analytic power and flexibility of the tool within which they are placed. For example, the centerline and cross-section must be able to move and resize themselves parametrically or via manual “grip point” operations, and the nodalization (both axial and circumferential) must be variable.

These challenges have been met in recent developments. Fortunately, the underlying flow solutions and heat pipe modeling methods were already in place: the new 2D pipe walls essentially automate access to those preexisting analysis features.

A good example of both nonlinear centerlines and noncircular cross-sections is provided by the racetrack heat pipes shown in Figure 9. In the electronics industry, low vapor pressure working fluids such as water are feasible, permitting the container to deviate from the traditional pressure-containing circular shape of aerospace ammonia heat pipes. Pipes with a racetrack cross-section (Figure 10) are produced inexpensively in large quantities. They have the benefit of needing no saddle since they present a flat side for mounting. They also take up minimal space, which is always at a premium in electronics packaging.

When modeling noncircular cross-sections, the user may draw an arbitrary closed curve using CAD drawing tools: lines, arcs, etc. Similar tools may be used to construct the centerline itself, whether the wall is 1D or 2D, the primary difference being that a centerline is not usually closed. The cross-section is then extruded along the centerline to create the shell representing the wall.

If the centerline has an angle (versus a smooth arc), the resulting wall surface will contain a mitered joint. If the centerline has an arc, the cross-section is swept through the arc. In both cases, the endpoint lines and arc tangents are

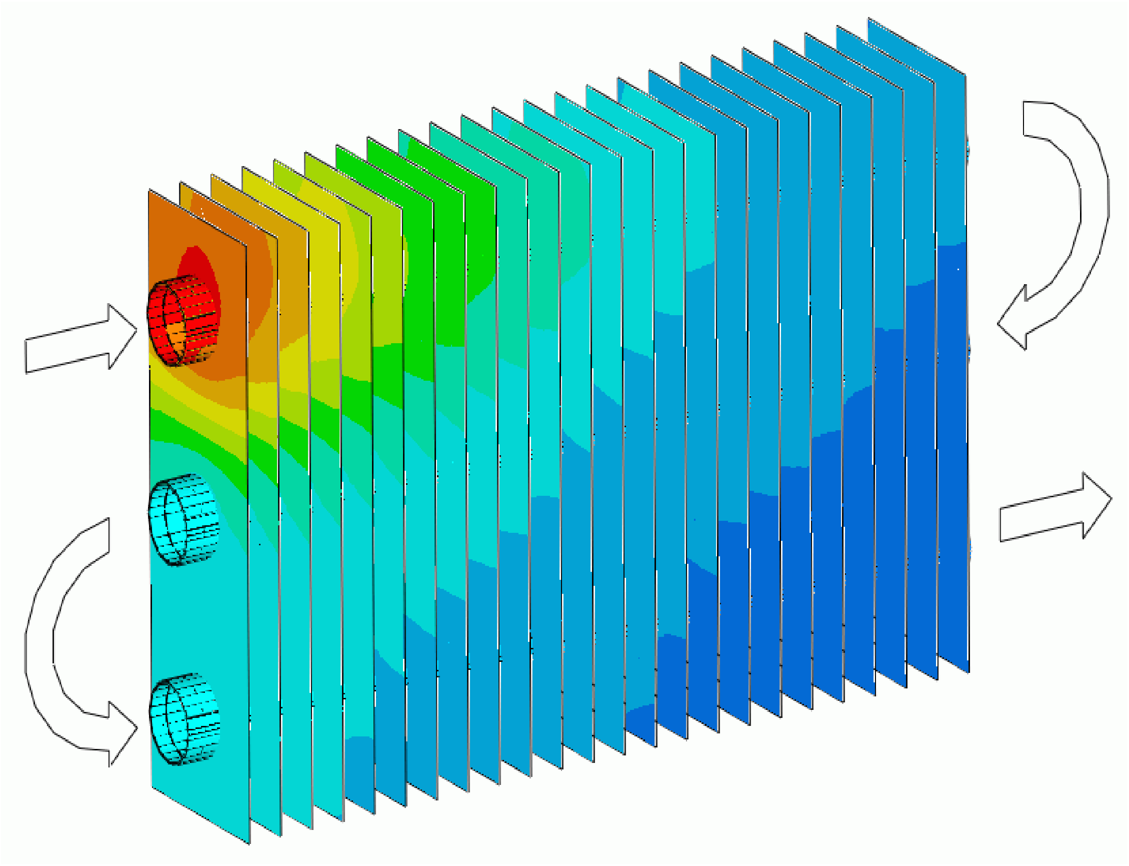


Figure 8: Air-cooled R134A Condenser: 2D FDM Pipe with 2D FEM Fins

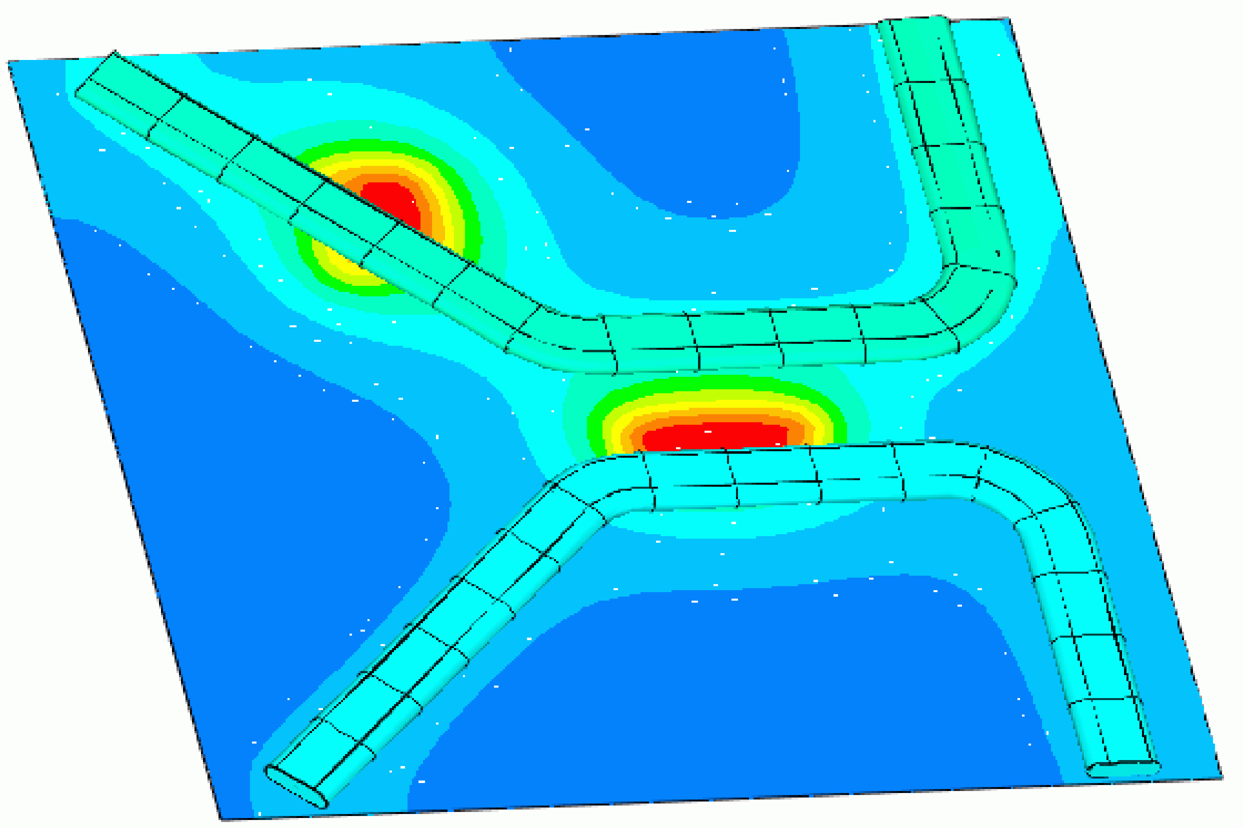


Figure 9: Racetrack-shaped 2D Heat Pipe Extruded Along Complex Curve, Bonded to FEM Surface

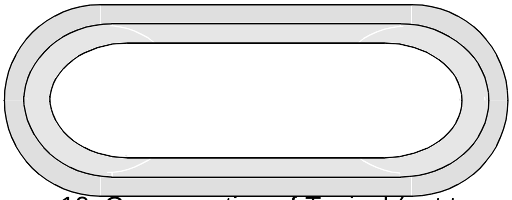


Figure 10: Cross-section of Typical (not to scale) Racetrack Heat Pipe

used to decide how the resulting 3D shape should appear. For example, such algorithms are responsible for “knowing” that the racetrack pipes in Figure 9 stay flat against the plate when the centerline bends.

These algorithms are re-executed if the centerline or cross-section are modified, whether by cursor-based click and drag operations or by algebraic variations of the underlying parameters, including *during* analysis runs (say, for example, for sizing or locating studies, see Ref 6).

CONCLUSIONS

Popular thermal management devices such as heat pipes, vapor chamber fins, loop heat pipes, loop thermosyphons, pumped single-phase coolant loops, spray coolers, and vapor compression cycle refrigeration loops are difficult to simulate using 2D/3D CFD techniques: 1D and quasi-2D network modeling techniques are much more appropriate. However, 1D flow modeling techniques were not previously compatible with the widespread used of 2D/3D thermal (conduction/radiation/capacitance) modeling software.

This paper has described a 1D flow modeling tool specifically intended to redress this gap in simulation technology, and has used brief examples to demonstrate the concepts involved. The speed of the resulting simulations enables higher-level tasks such as optimization, worst-case scenario seeking, automated calibration to test data, and reliability/sensitivity assessments via statistical design methods.

REFERENCES

1. T. Panczak et al, Thermal Desktop® User's Manual, www.crtech.com.
2. D. Johnson et al, “Novel Simulation Techniques for Design of Air-cooled Electronics,” IPACK2001-15523, July 2001.
3. S. Ring, Sinaps*Plus*® User’s Manual, www.crtech.com.
4. J. Baumann et al, “Nonlinear Programming Applied to Calibrating Thermal and Fluid Models to Test Data,” SEMI-THERM, March 2002.
5. B. Cullimore, “Dealing with Uncertainty and Variations in Thermal Design,” IPACK2001-15516, July 2001.
6. B. Cullimore, “Nonlinear Programming Applied to Thermal and Fluid Design Optimization,” ITHERM, May 2002.
7. SINDA/FLUINT User’s Manual, HEATPIPE and HEATPIPE2 utilities, www.crtech.com
8. J. Ku, “Operating Characteristics of Loop Heat Pipes,” SAE 1999-01-2007, July 1999.
9. J Baumann et al, “Steady State and Transient Loop Heat Pipe Modeling,” SAE 2000-ICES-105, July 2000.
10. J. Baumann et al: “Noncondensible Gas, Mass, and Adverse Tilt Effects on the Start-up of Loop Heat Pipes,” SAE 1999-01-2048.
11. J. Baumann et al, “An Analytical Methodology for Evaluating Start-up of Loop Heat Pipes,” AIAA 2000-2285.
12. Thermacore's Xeon™ Processor Cooler, www.thermacore.com/therma_sink.htm

DEFINITIONS, ACRONYMS, ABBREVIATIONS

CAD	Computer Aided Design
CCHP	Constant Conductance Heat Pipe
CFD	Computational Fluid Dynamics
FCHP	Fixed Conductance Heat Pipe (aka CCHP)
FDM	Finite Difference Modeling
FEM	Finite Element Modeling
FloCAD®	Fluid system analyzer in Thermal Desktop
LHP	Loop Heat Pipe
LTS	Loop Thermosyphon
NCG	Noncondensible Gas
RadCAD®	Radiation analyzer in Thermal Desktop
RADK	Radiation conductor (network element)
SINDA/FLUINT	Thermal/fluid analyzer from C&R Technologies
SINDA	Thermal side of SINDA/FLUINT
Thermal Desktop®	CAD-based FDM/FEM thermal modeling environment from C&R Technologies
VCHP	Variable Conductance Heat Pipe

CONTACT INFORMATION

C&R Technologies, Inc.
e-mail: ....brent.cullimore@crtech.com
e-mail: ....dave.johnson@crtech.com
URL: ....www.crtech.com
Voice: 303 971 0292
Fax: 303 971 0035