

An Innovative Modeling Approach to Thermal Management using Variable Fidelity Flow Network Models Imbedded in a 3D Analysis

2011-01-1048

Published
04/12/2011Hamish Lewis, Ben Zandi and Jeffrey Lewis
TES InternationalSatish Ketkar PhD
Wayne State University

Copyright © 2011 SAE International

doi:[10.4271/2011-01-1048](https://doi.org/10.4271/2011-01-1048)

ABSTRACT

Speed and accuracy are the critical needs in software for the modeling and simulation of vehicle cooling systems. Currently, there are two approaches used in commercially available thermal analysis software packages: 1) detailed modeling using complex and sophisticated three-dimensional (3D) heat transfer and computational fluid dynamics, and 2) rough modeling using one-dimensional (1D) simplistic network solvers (flow and thermal) for quick prediction of flow and thermal fields. The first approach offers accuracy at the cost of speed, while the second approach provides the simulation speed, sacrificing accuracy and can possibly lead to oversimplification. Therefore, the analyst is often forced to make a choice between the two approaches, or find a way to link or couple the two methods. The linking between one-dimensional and three-dimensional models using separate software packages has been attempted and successfully accomplished for a number of years. However, this coupling procedure involves a very tedious and time-consuming task of interfacing between the two packages made more difficult by the lack of access to the source code. Furthermore, there may be issues relating to overall convergence, as well as the convergence of each solver, which can lead to compromised accuracy. For a truly coupled approach modifications to the source code of the solvers are required.

This paper introduces a complete thermal modeling and simulation methodology that offers a variety of approaches for modeling complex systems and components, avoiding any of the unnecessary “overhead” associated with a single

modeling approach type. As an example, by allowing the combination of a full three-dimensional analysis in the radiator and the use of a network-based approach for pipes and pumps, the analyst can study the effects of orientation, fouling and related environmental conditions within the radiator and its effect on the complete system. The ability to combine these modeling approaches within a single system model allows one to employ the right tool for the job and avoid any unnecessary overhead associated with a single modeling approach or linking different packages. This greatly reduces the model development / analysis time allowing for the timely generation of results from which informed design decisions can be inferred.

A new software package, ADFlo, under development with the support of the US Army is used to demonstrate these benefits. This approach is applied to the thermal management of a liquid cooled Electronics Rack to demonstrate the ability to solve complex systems. The rack, boxes and the heat exchanger are all modeled in three-dimensions, with a one-dimensional flow network running through them. The flow network is complete with a pump and is tied to a one-dimensional airside model. The model is validated by experimental data.

INTRODUCTION

Today's military ground vehicles are filled with an increasing amount of electronic systems to satisfy various mission objectives. These electronic systems must operate in some of the harshest environments on earth, including the summer heat in the deserts of Iraq and the low temperatures of winter

in Afghanistan's high-altitude. Failure of these mission critical components is not an option. Among the critical electronic systems found onboard military vehicles are command, communications, control, intelligence, surveillance and weapon systems. Military officials are pushing the envelope with onboard electronics, requiring more processing power and more ruggedness in a smaller envelope for less cost. The added electronic subsystems are in turn, heavily taxing the thermal management and power distribution systems on many vehicles. The associated thermal issues have led to a requirement for liquid cooled electronics for the new compact electronics enclosures. As electronics improve over time, the ARMY has the ability to fit more computing power in a smaller envelope. Many times this creates a requirement for additional simulation to ensure that the cooling system is capable of handling the increased power density.

This simulation requires an innovative design and simulation tool for rapid thermal analysis and assessment of cooling system capability. This process could be used to design upgrades for cooling systems in existing vehicles and for the overall design of cooling systems in future military vehicles to meet established Army reliability and maintenance requirements. Generally, there are two approaches used for virtual thermal analysis of cooling systems. The first approach consists of detailed modeling of the system component(s) using commercially available Finite Element / Computational Fluid Dynamics (CFD) packages. Although this approach can yield very accurate predictions, it is not suited for rapid thermal management needs. Detailed model creation is a lengthy and time-consuming process requiring a high degree of expertise. Furthermore, the created models are representative of a specific cooling component and scenario; and thus, not suitable for rapid prototyping and "what-if" exercises. These simulations are extremely computer resource intensive (CPU and RAM). This approach is impractical for simulation of complex cooling systems comprising of several cooling components. The second approach involves the use of "Network" solvers (flow and thermal). This approach is efficient for quick prediction of the flow field in a variety of applications and is the primary solution methodology for several thermal analysis packages in the market. The problem with this methodology is that it may not be suitable for modeling some components. In these situations, the network solvers must be linked or coupled to solutions obtained using other methods. The coupling of different software packages, with no access to the source code, is at best a very tedious task and can lead to lower than necessary thermal margins, which unnecessarily handcuffs missions.

Our proposed solution provides a complete thermal management environment that offers a variety of approaches each ideal for modeling a type of cooling component/ scenario. This will include the ability to link and couple various model types, from simple "network" based models

using pre-defined system components from the component database, to more complex models involving a mixture of "networks" and full CFD/Computational Heat Transfer (CHT) based component modeling. As mentioned above, "network" modeling is the ideal approach for complex cooling system thermal analysis. Though simplified in nature, "network" based models capture the essential process in a thermal system and enables the analyst to rapidly characterize the system performance with a reasonable degree of accuracy. A full CFD/CHT based component model can readily be incorporated into the "network" model if a given component, such as an electronics box, requires a more detailed analysis. This mixing of modeling techniques has the advantage of applying detailed, and thus more computationally intensive, analysis only where needed. The interfacing of "network" and CFD/CHT based component models can readily be accomplished through boundary condition matching techniques.

METHODOLOGY

MODEL SETUP



Figure 1. Front of Rack Model

In this system, there are five LRU's (Line-Replaceable Units) electronics boxes mounted within a rack enclosure. This is typical of what is found in aerospace and some of the new military ground vehicles. The rack can be seen in [Figures 1](#) and [2](#). The heat dissipated in these boxes must be efficiently removed to maintain their functionality. The rack is liquid cooled and provides coolant to the LRU's found within it. The

model takes into account all conduction through the rack and the natural convection found within the rack.

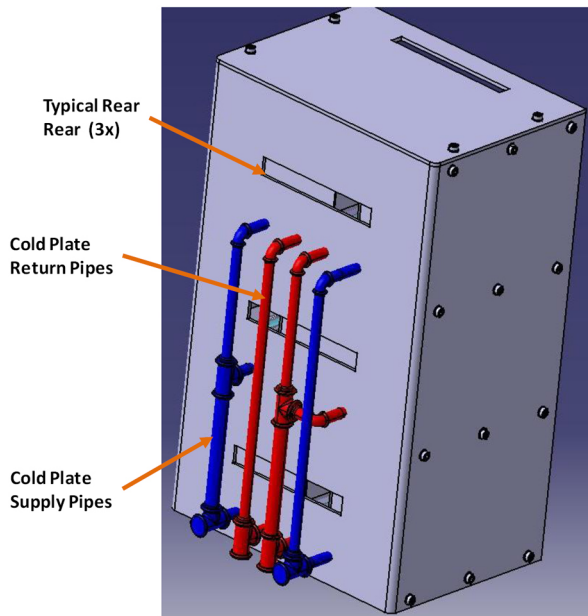


Figure 2. Back of the Rack Model

In order to simulate a realistic situation, we are treating three of the LRUs as “black boxes”. This means that the internal workings of the LRUs are not available to us (which is often the case in system level analysis). For the “black boxes”, we are modeling the external geometry and putting in a specified amount of heat. The LRUs will affect the temperature of the coolant, as well as the air temperature within the rack.

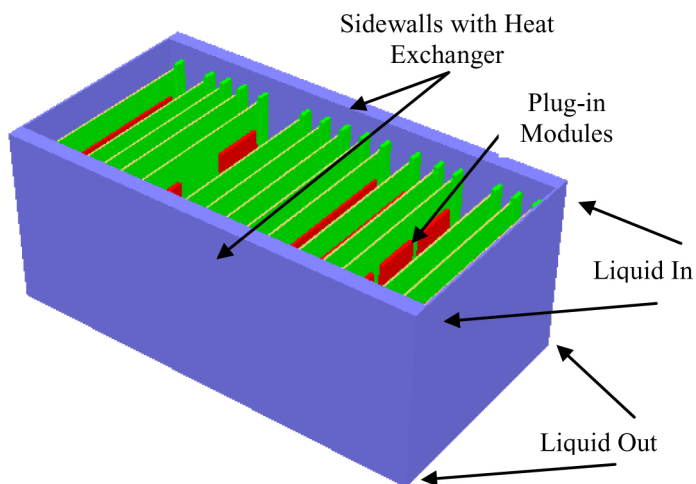


Figure 3. Model of Electronics Box Containing many Modules.

We modeled the interior of the center and largest LRU, which can be seen in [Figure 3](#). The electronic box is made up of one motherboard with plug-in modules that are cooled by

conduction of heat to the liquid cooled sidewalls [4]. Typically, in these applications, a large amount of heat dissipated by the electronic components must flow through the PCB's (with many copper layers to enhance in-plane conduction) to the chassis sidewalls. The heat is then removed by coolant through the channels of the sidewalls (called cold-plates). These channels include tabulators to provide turbulence to the coolant that increases the rate of heat transfer. The modeling of the turbulators in CFD is very time consuming, but since the coolant is modeled using a network approach, adding the turbulators to the analysis just requires modifying the Nusselt number and friction factor for the associated flow bars. The component junction temperatures within the LRU are controlled by the effectiveness of the conductive heat path (i.e., component-to-board interface, conduction through PCB, conduction through clamps from PCB to chassis) and the efficiency of the coldplate.

The main difficulty in modeling these problems lies in the modeling of exchangers. A numerical CFD model of the box that incorporates the details of the heat exchangers would be prohibitively large and impractical for most applications. In this study, we will use a fluid network heat exchanger model that is coupled with the main thermal / CFD solution scheme. The proposed simplified heat exchanger model can take advantage of an existing library of heat exchanger performance data, or alternatively, it can use existing heat exchanger performance curves. The ultimate utility of this methodology is to allow quick and accurate design decisions at system-level during package development by enabling the analyst to construct models and perform system-level thermal (and CFD) analyses in a matter of a few hours, rather than a few days (or weeks). The effort required to adequately model the transient response of a realistic air to liquid heat exchanger by CFD is often prohibitive despite the prevalence of large clusters. The transient response can be accurately handled through an RC network, however short transient (less than a few seconds) will not be as accurate for the heat exchanger model, unless empirical data is available to that level of detail.

The second LRU that we modeled is a high voltage motor controller, seen in [Figure 4](#). This is made up of many boards, as well as a group of IGBTs that all give off a substantial amount of heat. The current paths through the boards are modeled and the heat that is created by the current flowing through the traces is calculated using a built in voltage solver. It is critical that the voltage solver be coupled to the thermal solver because the resistance through the leads and traces increases with the temperature. This in turn increases the heat generated by the current. The LRU has a liquid cooled coldplate going through its center with many of the components generating the most heat thermally connected to it.

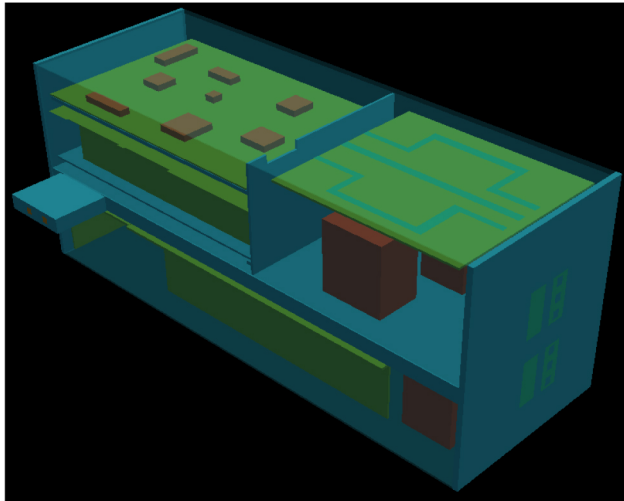


Figure 4. High Voltage Motor Controller

The final component that we modeled in 3D was the heat exchanger, seen in [Figure 5](#). Though the liquid side of the heat exchanger was modeled through a network approach, we choose to model the solid and airside through 3D CFD. The entire heat exchanger could easily be modeled with a network approach, but often if you are looking to optimize ducting or placement, a 3D model provides a greater insight. The model consists of an aluminum box, with many fins going through it. The liquid travels through the fins in coils (which is modeled through a network approach). The hot coolant flowing through the network model heats the fins, which are then cooled by the air being forced through them. The velocity of the air is modeled with a fan curve, showing airflow versus pressure drop.

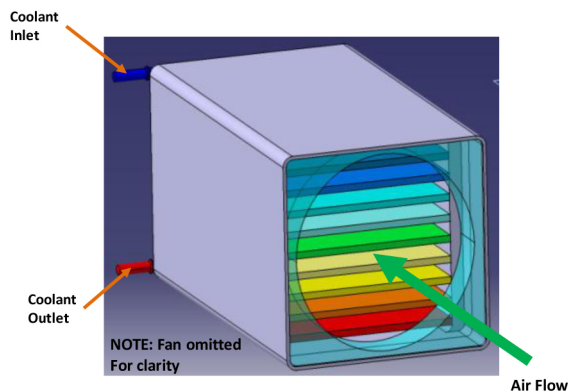


Figure 5. 3D Model of Heat Exchanger

SOLVER METHODOLOGY

In order to obtain all the necessary detail of our analysis, without any extraneous modeling, we used a variable fidelity approach. The following approach was taken for thermal/flow modeling and simulation of the entire system:

- Network Modeling

- Bernoulli flow bars were used for the piping between the rack, pump and the heat exchanger, and the pump itself
- NxGen flow bars consisting of a series of nodes were used for the flow passages through the cold plates and the liquid coolant side of the heat exchanger.

Full three-dimensional CFD/CHT was employed for the component boxes, rack enclosure, cold plates (excluding the flow passages), and ambient air-flow through both the rack enclosure and air side of the heat exchanger. It is in the aluminum cold-plates and the heat exchanger where the embedding and coupling, respectively, of network based and three-dimensional CFD/CHT based modeling is highlighted.

FLOW NETWORK

One of the design goals of the NxGen network solver was to formulate a network modeling approach that would have maximum flexibility and applicability while still maintaining ease of application for the analyst. The majority of network solvers available are of the Bernoulli type, with respect to flow related calculations, in that velocity and pressure are related to some type of loss coefficient via an algebraic expression [1]. However, the underlying assumptions involved in the derivation of the Bernoulli equation are quite restrictive and thus impose many limitations with respect to its applicability. Furthermore, some of these network solvers have no provisions with respect to thermal considerations, and if so they are limited to simple first law type analysis.

Since the NxGen network solver is based on a full one-dimensional CFD/CHT type analysis, it overcomes these limitations and thus affords a far greater range of applicability. It should be noted at this point that in some situations, a Bernoulli type flow network is quite adequate and in others may be the only choice due to limited information regarding the component and therefore it remains an integral part of the thermal simulation platform. However, the rich feature set and wide range of applicability make the NxGen network modeling the tool of choice. As indicated earlier, the basis of the NxGen network solver is a one-dimensional CFD/CHT formulation. Thus, the development of its governing equations begins with the fundamental conservation laws for mass, linear momentum, and energy. We recall that for a fixed, non-deformable control volume these equations take the following form:

Conservation of Mass:

$$\iiint_{CV} \frac{\partial}{\partial t} \rho \, dV + \iint_{CS} \rho (\vec{V} \cdot \vec{n}) \, dA = 0 \quad (1)$$