VARIABLE FIDELITY METHODOLOGY FOR THERMAL BATTERY MODELING

Hamish Lewis, Dr. Ben Zandi, Gareth Lewis
TES International LLC
1711 Larchwood
Troy, MI, USA, 48083
Phone: (248) 747-8994
Email: hlewis@tesint.com

Dr. Satish Ketkar
Wayne State University
Detroit, MI 48202
Phone: (248) 879-9347
Email: ketkar@yahoo.com

ABSTRACT

Cell temperature is critical to the life and performance of a battery pack, therefore accurately modeling the complete thermal system is important. Traditional approaches use complex 3D heat transfer and CFD or use a 1D network solver for quick prediction of flow and thermal fields. The 1D and 3D models are linked to get an overall system model. However, this coupling procedure can be tedious and time-consuming. This paper introduces a complete thermal modeling methodology that offers approaches for modeling thermal management system combining the benefits of 1D and 3D models in a single system greatly reducing the model development and analysis time. A new software package, ADFlo, developed with the support of the US Army is used to demonstrate these benefits. This approach is applied to the thermal management system for a battery pack where flow balancing of the coolant has a significant impact on the performance of the system. Properly balanced flow will create an even temperature distribution increasing the performance and life of the system. This will be seen on a 3D solid model of the battery. The analyst can use this to study the effects of modification to the cold plate to ensure an even temperature.

KEY WORDS: flow network, analysis, cooling, system, CAE, automotive, 3D, pressure

NOMENCLATURE

1D	One Dimensional
3D	Three Dimensional
AC	Alternate Current
C	Celsius
CFD	Computational Fluid Dynamics
CHT	Conjugate Heat Transfer
DT	Temperature Difference
Hp	Horsepower
HEV	Hybrid Electric Vehicles
HVAC	Heating, Ventilation and Air Conditioning
IGBT	Insulated Gate Bipolar Transistor
ICE	Internal Combustion Engine
kW	Kilowatts
MGU	Motor Grinder Units
OEM	Original Equipment Manufacturer
PEV	Plugin Electric Vehicles
SIMPLER	Semi-Implicit Method for Pressure-Linked Equations

INTRODUCTION

Due to the increasing popularity of both HEV (Hybrid Electric Vehicles) and PEV (Plugin Electric Vehicles) the need for more accurate and faster analysis of the cooling systems responsible for these vehicles is increasing. Due to the importance of battery cell temperature on the life and performance of a battery pack system, it is critical to accurately model the complete thermal system. 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. However, it is not always possible to do a complete three-dimensional model due to lack of time and/or data.
2. Using a one-dimensional (1D) simplistic network solver (flow and thermal) for quick prediction of flow and thermal fields. Although this provides the modeling and simulation speed needed, sacrificing accuracy and can possibly lead to oversimplification.

The linking between 1D and 3D models using separate packages has been used for a number of years. However, this coupling procedure can be tedious and time-consuming without access to the source code. Furthermore, there may be issues relating to keeping track of overall convergence, which can lead to compromising 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 full thermal management systems and components, avoiding any unnecessary "overhead" that would be associated with a single modeling approach type. Combining these modeling approaches within a single system model, greatly reduces the model development and analysis time allowing for the timely generation of results from which informed design decisions can be inferred.

A new software package developed with the support of the US Army is used to demonstrate these benefits. This approach is applied to the thermal management system for a battery pack. The cooling medium and flow balancing of the coolant has a significant impact on the performance of the system. Properly balanced flow will create an even temperature distribution increasing both the performance and life of the system. This will be seen on a 3D solid model of the battery cells, complete with full 3D CFD of the cooling channels within the cells. The CFD will be strongly coupled to a flow network-based approach for the liquid flow through the pipes between the cells, as well as heat exchanger and pumps. The analyst can use this to study the effects of modification to the cold plate to ensure an even temperature.

MODEL DESCRIPTION

In order to demonstrate the capabilities of the software, ADFlo™, we have created a thermal analysis model of the Chevy Volt cooling system. The Chevy Volt cooling system is one of the most complex systems found in the history of automotive. This is because not only does it have two separate power trains, but due to the fragile thermal nature of batteries, the Volt battery must be kept within a narrow thermal range below 40°C[4]. This means that the coolant is used not just to cool, but also heat the battery. In Figure 1, we see the thermal management systems of the Volt Battery System.

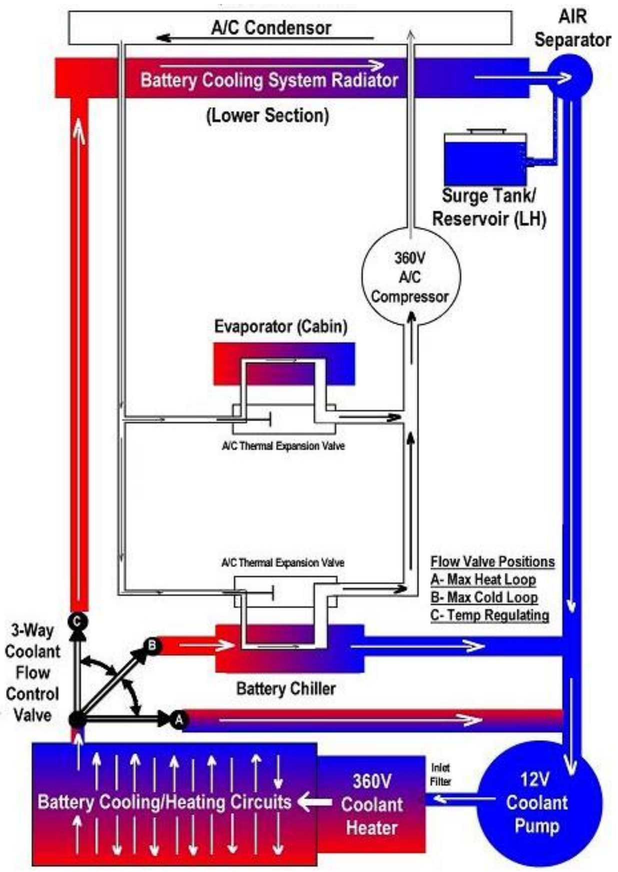

Fig.1 Battery Thermal Management and HVAC System [1]

In order to maintain the temperature of the battery within the narrow optimum range, the cooling system is equipped with multiple thermal circuits. The 3-way electronic flow valve sends the coolant between a radiator for use of a normal day while driving, a heater for cold starts and a battery chiller connected to the HVAC system of the Volt. The chiller is used for hot day situations and conditions when the battery must be cooled while the vehicle is at rest.

With an electric drivetrain come high powered electronics. The power control modules are made up of high powered IGBTs and Diodes that require liquid cooling. Although liquid cooled electronics have been uncommon in automotive applications, they have been the standard in aerospace for years. The electronics cooling circuit is shown in Figure 2. It is a very basic cooling circuit, using a liquid to air heat exchanger (radiator) to maintain a coolant temperature below 70°C.[4]

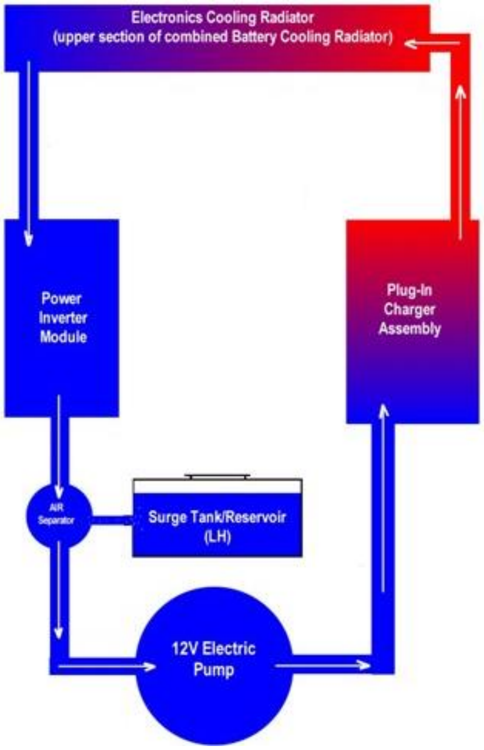

Fig.2 Power Electronics Cooling System [1]

Since the Chevy Volt still has an Internal Combustion Engine (ICE), it has a standard cooling system that has been found in automobiles for decades. This is seen in figure 3. The only difference between this system and the standard cooling system is that because the engine is not always running, it cannot be relied on to heat the passenger compartment. Therefore, a 360V electric heater is also installed in the circuit for conditions where more heat is required than is given off by the ICE. The water pump is also electric instead of belt driven because the engine is not always running. This allows for a higher level of optimization, but also adds complexity.

The electric drive unit cooling and lubrication system is designed to maintain the internal temperature of the transaxle used in the Chevy Volt. The electric drive unit propels the Volt using electric power, as well as generating electricity to maintain the battery state-of-charge. It is made up of two high power motor generator units (MGU-A is 58 kilowatts and MGU-B has a peak of 116 kilowatts) so there is a need to actively disperse the heat during operation. [1]

The pressure within the transmission fluid system is provided by an electrical motor/pump assembly within the transaxle, but it's also driven by a mechanical pump to ensure the transmission fluid pressures and flow are present whenever the engine is running.

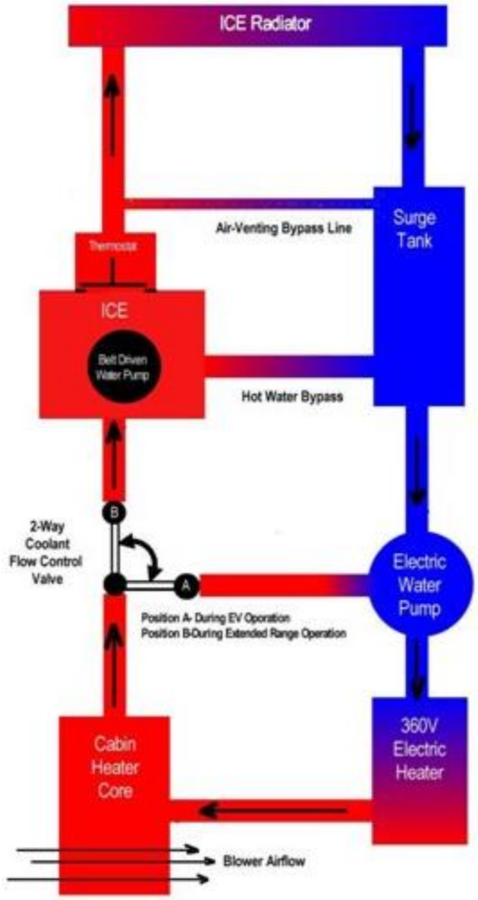

Fig.3 Engine Cooling and Cabin Heating System [1]

An external transaxle cooler outlet fitting directs fluid under pressure into a high pressure line and to the transmission fluid heat exchanger (radiator/cooler) mounted between the engine cooling radiator and air-conditioning condenser. Transmission fluid circulates through the cooler tubes as airflow across the radiator withdraws heat from the fluid. An outlet fitting from the transmission cooler/radiator then directs the cooled Dexron fluid back into the transaxle via the return line. There is a transmission cooler fluid bypass device at the IN/OUT fitting of the cooler so that in the event of a restricted cooler (due to debris or extremely cold temperatures) the bypass valve would open and redirect the fluid back to the transaxle return fitting.

Although the focus of the paper is on the battery cooling, the reason we are modeling all four of the cooling systems found within the Volt are that all the radiators are found in series. The diagram of the radiators and how their relative locations can be seen in figure 4.

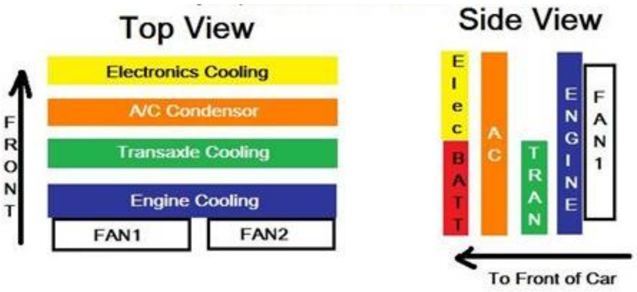

Fig.4 Chevy Volt Radiator Setup [1]

For this paper all the cooling systems other than the battery cooling system were roughly modeled due to a lack of information regarding their inner workings, but their effect on the battery was accurately captured. However, if necessary, all circuits could be modeled with the same detail as the battery cooling system allowing for a full vehicle thermal optimization within one system model.

One of the most critical goals of battery design is to ensure an even temperature distribution across each individual cell and the battery as a whole. For this paper, we have modeled the battery in three dimensions (3D), so that we can see the actual temperature distribution across each cell. The battery is made up of 288 cells. It is cooled by 144 thin cooling fins, with a cell on each side of the fin.[3] The cooling fin is seen in figure 6. It is both highly conductive to help balance the temperature distribution and extremely thin to keep the size and weight of the battery down. It contains 4 micro channels in parallel, and all fins are in a parallel flow path to help control the temperature throughout the battery evenly.

The heat dissipated from the cells is modeled as a temperature dependent function. As temperature increases, the amount of heat dissipated from each cell actually decreases, unlike with trace heating, this helps create a uniform temperature. The function is spread evenly throughout the cell, though the hotter regions will be producing less heat.

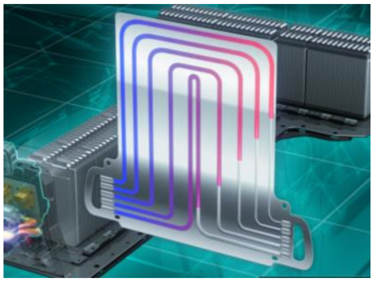

Fig.5 Battery Cooling Fin [2]

Although the solid of the battery is modeled in 3D, the coolant flow will all be modeled using our flow network solver. This is the perfect tool because all flow is through well-defined channels. The flow network model is embedded in the 3D model, so that the heat transfer between the two models is seamless. The 1D and 3D analysis are run within the same solver suite, so that they are strongly coupled throughout the simulation.

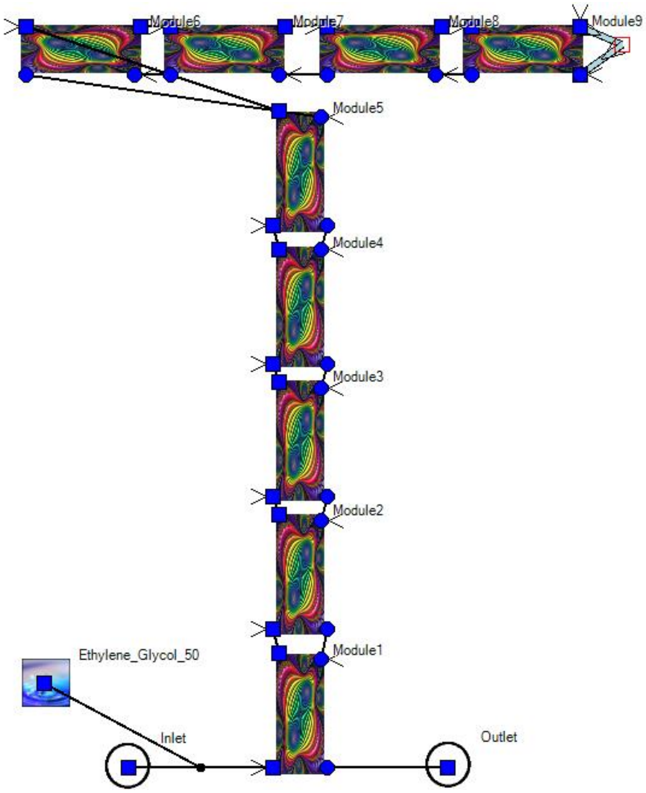

Fig.6 Flow Network Battery Model

The network battery model is seen in Figure 6. The battery is made up of 9 modules that are all identical. LG Chem, the battery manufacturer, created the module structure so that OEMs could easily tune the battery to the required power and capacity. Each module is made up of 16 cooling fin assemblies (32 battery cells) as shown in Figure 7. [3]

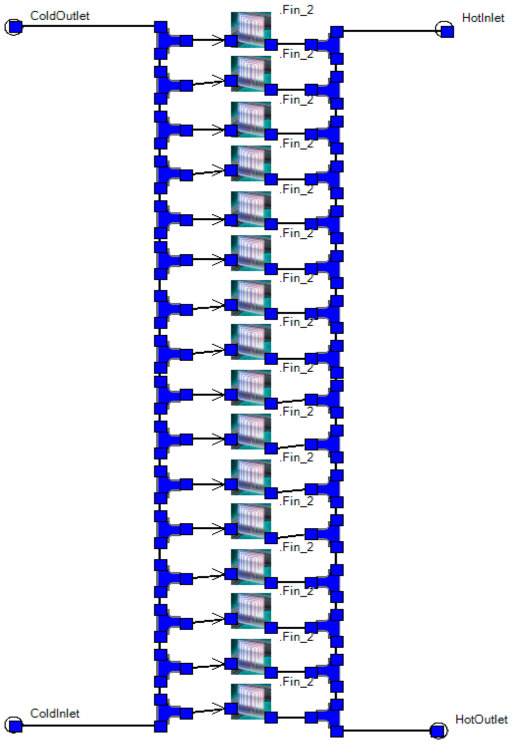

Fig.7 Module Model

The 3D representation is shown in figure 8. It is a simplified model of the CAD model also shown. Because the modules are covered with a thick plastic cover (seen in transparent blue) that protects it from the underhood environment, we have ignored thermal loses to the environment for the purpose of the paper. All the heat loss is through the coolant.

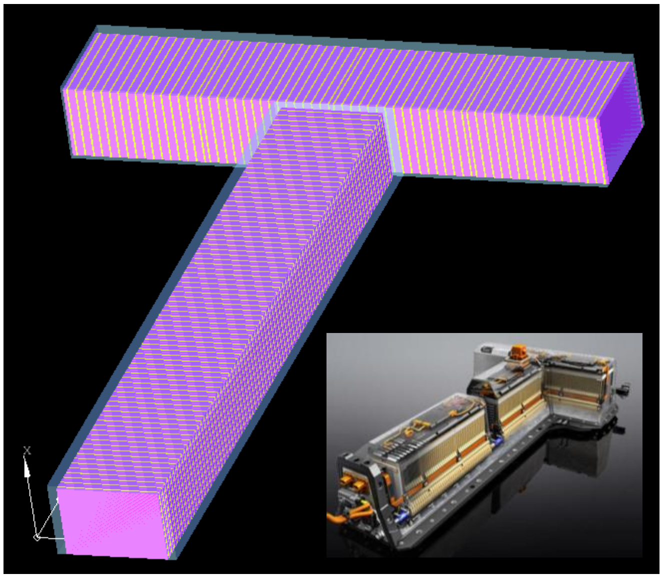

Fig.8 3D Thermal Model of Battery beside CAD model [2]

SOLVER METHODOLOGY

Speed and accuracy are critical factors for a good solver. A variable fidelity modeling approach is one way of accomplishing this. This is more important as the complexity and size of the models increases. Our system thermal/flow simulation solver includes two network modeling methodologies integrated with three-dimensional Computational Fluid Dynamics (CFD) if so desired, all in a single software package.

- A Network Model is used for the overall system model.
    - Areas requiring a less rigorous treatment, such as the piping between the rack, pump and the heat exchanger, and the pump itself, used Bernoulli flow bars.
    - NxGen flow bars consisting of a series of nodes were used where a somewhat greater degree of efficacy was beneficial. The flow passages through cold plates or the liquid side of the heat exchanger are examples.
- Full three-dimensional CFD/Computational Heat Transfer (CHT) was used for critical parts of the system. This included: component boxes, rack enclosure, cold plates (excluding the flow passages), and the ambient air-flow through the rack enclosure and air side of the heat exchanger. The embedding and coupling of network based and three-dimensional CFD/CHT based modeling is highlighted in the aluminum cold-plates and the heat exchanger.

FLOW NETWORK

The underlying assumptions used in Bernoulli type network solvers restrict the applicability of the approach. Flow velocity and pressure are related to a loss coefficient through an algebraic expression (1) and they are limited to simple first law analysis with respect to thermal issues if at all.

The NxGen network solver is a full one-dimensional CFD/CHT type analysis. It has a greater range of applicability than the Bernoulli approach. A Bernoulli type flow network may in some instances be adequate or may be the only choice due to limited information and is therefore an integral part of the thermal simulation platform. However, the NxGen network modeling is the tool of choice due to its extensive features and broad applicability.

The governing equations of the NxGen Network solver are the laws of conservation of mass, momentum and energy. For a fixed, non-deformable control volume these equations take the following form[5]:

Conservation of Mass:

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

Linear Momentum

$$\sum F_{X_L} = \frac{\partial}{\partial t}\left\{\iiint_{CV} \rho U \, d\mathcal{V}\right\} + \iint_{CS} \rho U(\vec{V}\cdot\vec{n})\,dA$$

Conservation of Energy

$$\dot{Q} - \dot{W} = \frac{\partial}{\partial t}\left\{\iiint_{CV} \rho e \, d\mathcal{V}\right\} + \iint_{CS} \rho e(\vec{V}\cdot\vec{n})\,dA$$

We ignore kinetic and potential energy effects as we assume that the internal energy is the total energy within the control volume. The above equations will also be used to ensure that conservation is not violated at the interface of the three approaches described above; namely the Bernoulli networks, NxGen networks, and full three-dimensional CFD/CHT models

Consider the conservation of mass between a network based component model and the outlet of a three-dimensional CFD/CHT component model as shown below.

If one assumes that the node at the interface cannot store any mass and that density is not a function of time, mass conservation applied to the interface node becomes;

$$\iint_{CS} \rho\,(\vec{V}\cdot\vec{n})\,dA = 0$$

or

$$\iint_{CS} \rho\,(V)_{CFD}\,dA = \rho(\,VA_{CS})_{NET}$$

The integral on the left hand side can be evaluated via any convenient quadrature method. Similar expressions for linear momentum and energy conservation can be developed at the interface node to complete the coupling.

A Network element represents a computational segment of a component modeled using either the Bernoulli or NxGen network modeling approach. A Bernoulli element has only two computational nodes within the element representing the endpoints. A NxGen network element also contains two endpoint nodes in addition to internal “sub-nodes” due to the CFD/CHT methodology. This “meshing” of the network elements is automatic and invisible to the user. Once meshing is complete, the network element contains a number of “sub-elements”, to which we apply the conservation equations. Making the “sub-elements” differential in size, shown below, the general conservation equations applied to this differential element yield the desired differential forms.

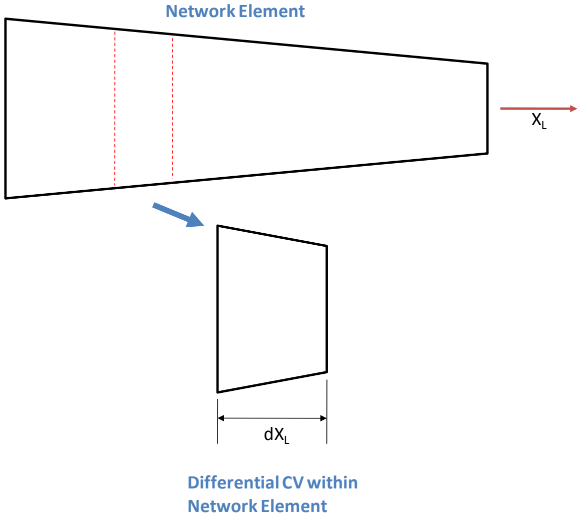

Fig.9 NxGen Network Differential Element

The equations are non-linear and are implicitly coupled as density and enthalpy are dependent on both temperature and pressure, and viscosity and thermal conductivity are dependent on temperature. There is no explicit equation for the determination of the pressure field. Lastly, two equations of state are required to relate the density and enthalpy to the pressure and temperature. There are several standard solution methods for the first two issues, but it is difficult to develop expressions for the Equations of State that are valid for a variety of substances over a wide range of pressures and temperatures, and include the effects of phase change etc. A "steam tables" type approach is used with table look-ups to relate temperature, pressure, enthalpy, and density. This is different from most CFD/CHT implementations as there are no simplifying assumptions about the state properties of the materials. This is integral to our goal of maximum flexibility and applicability. This approach also allows the analysis of liquid-vapor phase change such as might be encountered in vehicle cooling systems and allows the modeling of vehicle air conditioning systems as part of the overall cooling system model.

The solution method is a modified version of the standard SIMPLER algorithm (Semi-Implicit Method for Pressure-Linked Equations – Revised) which addresses the pressure field calculation and pressure linkage issues.

A network element within a NxGen model of a component is sub-divided into some number of "sub-elements". Each of these "sub-elements" has a related "sub-node" which lies at its geometric center. These "sub-nodes" represent the locations where the dependent variables, i.e., velocity, pressure, temperature etc., will be computed by the NxGen solver using the finite volume approach. One of the advantages of the finite volume discretization approach is that even when coarse grids are employed conservation is always satisfied. Figure 8 below, shows the typical layout of a NxGen network element showing the arrangement of "sub-nodes/elements" along with a detail of a grid-point cluster utilized in the discretization process.

Following the standard finite volume approach to the integration of the differential governing equations and utilizing the methods outlined in the SIMPLER algorithm for development of the equations required therein the following set of algebraic equations results:

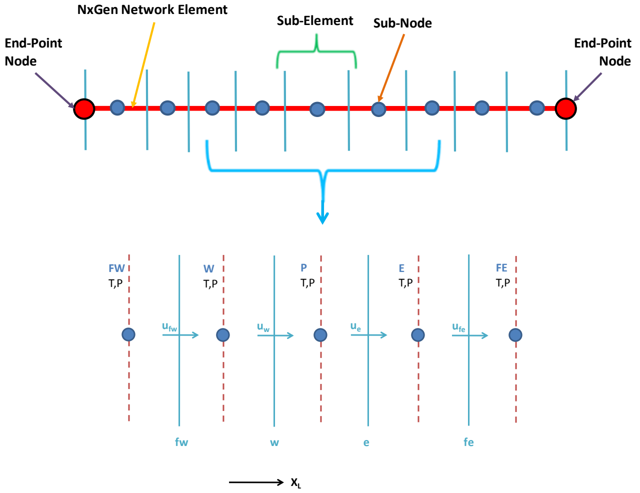

Fig.10 NxGen Network Elements

We have previously discussed the interfacing of the network-based models with 3D models within the same system and no further discussion is needed. The interfacing between NxGen and Bernoulli models is very straightforward in that they are both one-dimensional representations and thus the conservation equations at the interface nodes become trivial to apply. With respect to the overall solution scheme, it is envisioned that a hierarchical structure will be employed. At the start of any given iteration within an overall system model solution, the full three-dimensional CFD/CHT portions, i.e., those components modeled in this fashion, will be solved first utilizing all the latest system data available. Once these components have been resolved, and related system parameters updated, the NxGen network elements, along with any necessary interfacing calculations, will be solved again utilizing all the latest system data. Lastly, the Bernoulli network elements will be solved again using all the latest system information and related interfacing calculations. This process will continue until convergence of the overall system model has been reached. In the case of a steady-state analysis, once convergence has been reached the simulation is complete. For a transient simulation, once the current time point has converged, the time is advanced, boundary conditions are updated, and the solution process begins again utilizing the latest values as an initial guess for the new time point. This process continues until a given end time is reached.

3D SOLVER

The prediction of the heat transfer process requires understanding the values of the relevant variables (temperature, velocity, pressure, etc.) throughout the domain of interest.

(Energy Equation)

In the solid regions, the energy equation reduces to the heat diffusion (conduction) equation [6]. This equation in its most general form is:

$$\rho C_p \frac{\partial T}{\partial t} = \left\{ \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) \right\} + q'''$$

Note that the volumetric heat generation term, q′′′, represents total heat dissipation in element. The solution for the above equation depends on the physical conditions existing at the boundaries of the system (boundary conditions) and on the system's conditions existing at some initial time (initial conditions).

In the fluid region, it is necessary to solve a more general form of the energy equation to account for the convective effects.

$$\rho C_p \left(\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z} \right) = \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial x}\right) + q'''$$

Here, u, v and w represent the velocity components in x, y and z directions. The values of these components are not known and they must be computed by solving the continuity equation (conservation of mass) and the momentum equations (Navier-Stokes) equations.

For a Newtonian fluid with constant properties, these equations are below.

(Continuity Equation)

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

(x-Momentum)

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \right) = -\frac{\partial P_d}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right)$$

(y-Momentum)

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial P_d}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$

(z-Momentum)

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial P_d}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)+\rho g\beta(T-T_a)$$

These equations apply to both laminar and turbulent flow conditions. For turbulent flow, however, Reynolds averaging is used to decompose the variables into a mean and a fluctuating component. This leads to the creation of additional source terms (the so-called “Reynolds Stress” terms) in the momentum equations and one or more additional conservation equations for turbulence variables (i.e., k and Є equations). The complete set of equations must be solved simultaneously for temperature, pressure and three velocity components (and the turbulence variables if needed). The method used in this paper is based on “control volume” approach utilizing Patankar’s SIMPLER (Semi-Implicit Method for Pressure Linked Equations – Revised) algorithm[7].

The software used for this analysis is a finite volume package that incorporates the most important features for electronics cooling analysis. One of the key features is the use of coupled thermal/electrical algorithms in the solution. Thus, the thermal problem is solved simultaneously with the electrical field. This is important because the resistance of an electrical circuit varies with temperature that then affects the voltage and current fields in the circuit and the heat dissipated is a function of the current and resistance in each part of the circuit. The use of coupled thermal/electrical allows for a local solution which results in a much more accurate point reading of temperature than would otherwise be possible. The solution method involves many iterations and the electrical properties and thermal properties are both re-calculated during each iteration. This coupling becomes of greater value with the increasing current density of the electronics used in many areas. The increased accuracy and solution speed afforded become invaluable.

The package has computational fluid dynamics capability, although it can be run either with or without this feature turned on. A patented radiation solver is also incorporated. In many cases, radiation can be a more significant contributor to heat transfer than natural convection. The blocking technique used allows a much more efficient solution than would otherwise be possible.

VARIABLE FIDELITY

The combination of these approaches provide the analyst with several choices as to how to model any particular component within a cooling system given the degree of detail desired, complexity of the given component, and component data available for modeling. With respect to complex components such as heat exchangers, i.e., radiators, oil coolers, etc., the analyst can choose to implement a full three-dimensional CFD/CHT analysis to study the effects of orientation, fouling and related environmental conditions. For simpler components, such as coolant pumps and interconnecting piping/tubing, the analyst may choose to model these utilizing a network approach, either Bernoulli or NxGen, since the expense, with respect to modeling and analysis time, of the added detail may not be warranted. Thus, the ability to combine these modeling approaches within an overall system model, allows one to employ the “right tool for the job” and avoid any un-necessary “overhead” associated with a single modeling approach. This will greatly reduce the model development / analysis time allowing for the timely generation of results from which informed design decisions can be inferred. To further support this rapid model development cycle, the analyst will also have at their disposal, via the customized GUI, a user extensible database of pre-defined component models that can simply be “dragged” into a system model. These pre-defined models can be based on any number of approaches including network modeling, three-dimensional CFD/CHT modeling, test data, and/or manufacturer’s data.

TEST CASE

For the paper, we are using results from a hot day (50 C ambient) continuous operation, in which the engine is charging the battery and providing 40 kW (53.6 Hp) continuous mechanical power and the battery is providing a full continuous power of 45 kW (60.4 Hp).

The vehicle is assumed to be driving at 70 mph. The external air flow condition is started from 1 meter before the vehicle at this condition, and the air is allowed to pass through the underhood compartment (going through the series of radiators) or pass above or below the vehicle. The airflow is modeled using a flow network approach, with all the radiators being defined by existing components within ADFlo™ that use published correlations for determining the pressure loss coefficients for the airside and the thermal resistance from the liquid to solid to air.

As the air flows through each radiator, it picks up heat which ties all the thermal cooling systems together. Whenever designing the heat exchanger for a system, it is always important to take into account all the other systems it may affect through either reduced airflow or an increase air temperature.

Even though the radiator for the battery cooling system is the first in the series, since the external air temperature is above the required temperature of the battery cell, it will be required for the system flow valve to divert the flow through the Battery Chiller, which is a liquid to liquid heat exchanger connected to the AC loop. The HVAC system will be continuously on, to help maintain the battery and the flow valve will make sure that the coolant does not get too cold by switching between the chiller and the bypass. The control algorithm is embedded in the model, so if required the same model could be used for testing cold conditions or the charging scenario.

Results & Findings

Our initial goal was to run a baseline of a hot day condition. For our baseline we choose a diameter of 1 mm for each channel. In figure 11, we see a temperature plot of the battery cells at the front of the battery and the cell at the back of the battery that is last in the cooling path. Both cells have an internal temperature difference of less than 3°C and the difference between the hottest and coldest cell is also well below 3°C.

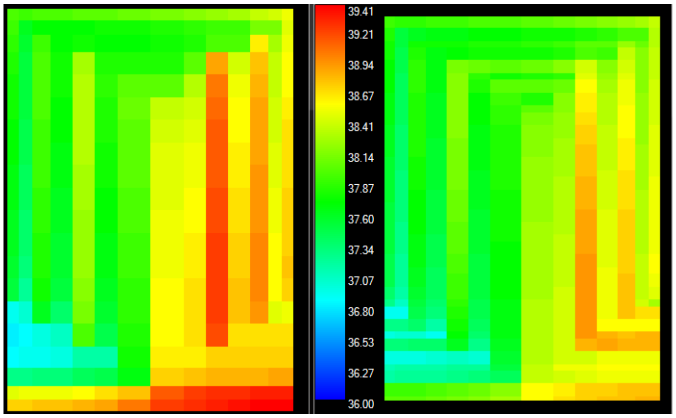

Fig.11 Temperature Profile of Last Cooling Fin on the left and the 1st Cooling Fin on the right

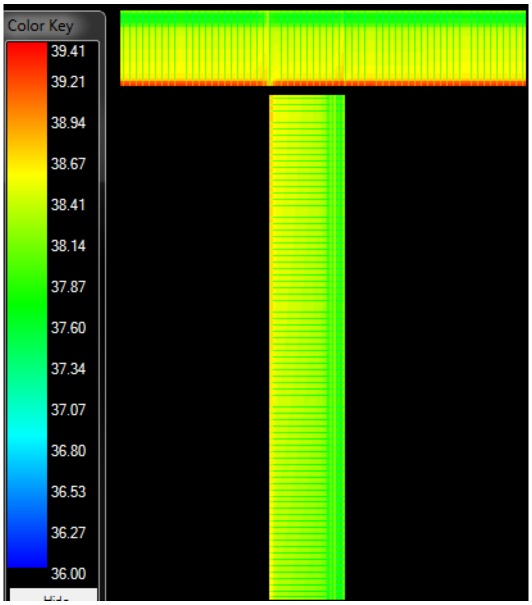

Fig.12 Cut plane of the center of the Battery Cells

In figure 12, we see the temperature of the battery in a cut plane look through the center of the thickness of the battery. Figure 13 shows how the temperature changes from the first to the last cell.

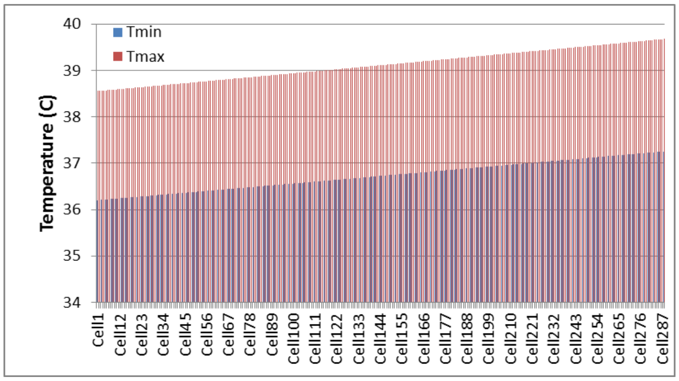

Fig.13 Plot of the Minimum and Maximum Temperature of each Cell

In figure 14 and 15, we can see the fluid temperature for both the first and last cell.

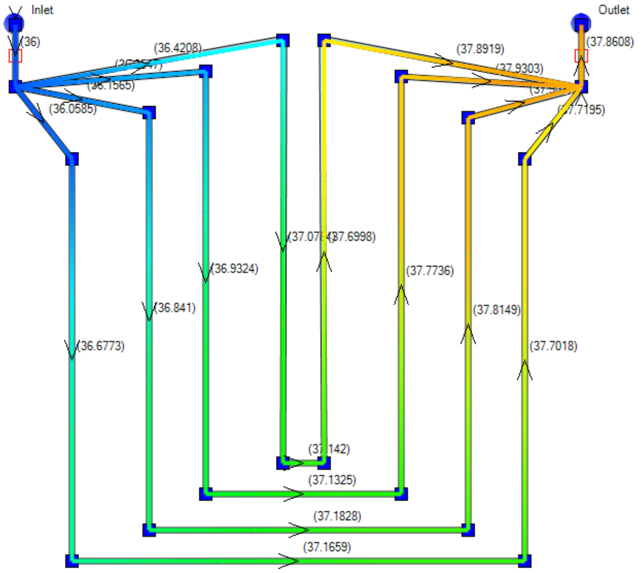

Fig.14 Plot of Fluid Temperature of First Cooling Fin

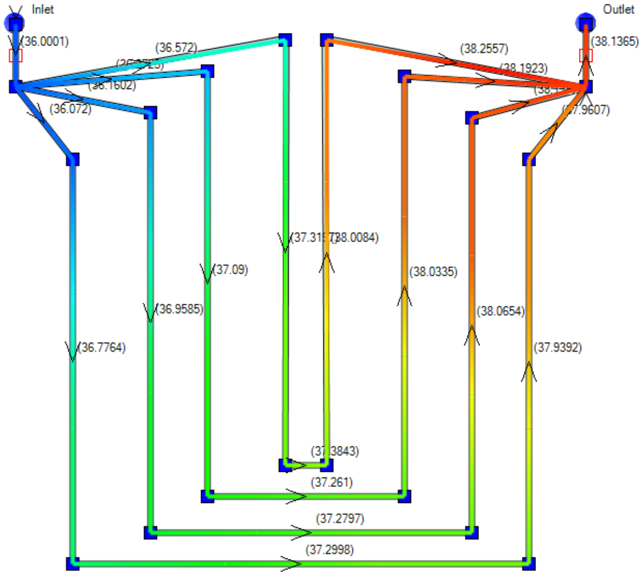

Fig.15 Plot of Fluid Temperature of Last Cooling Fin

In the following figure 16, we see how the flow rate across each fin changes through the battery. As you can see, the flow is almost the same through the first fin as it is through the last. This helps to provide the constant temperature required.

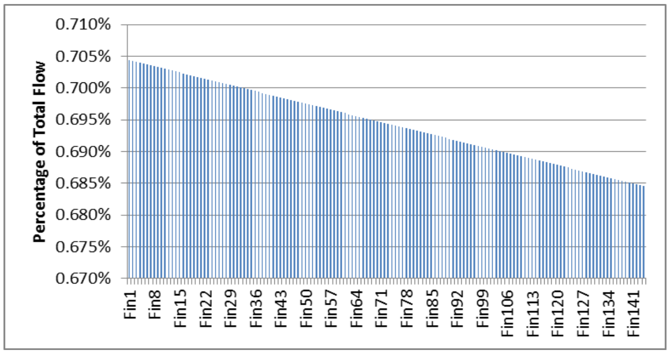

Fig.16 Percentage of Total Flowrate through each Cooling Fin

We are now ready to see if there is room to optimize the cooling fin channels. We optimized the fin to find the perfect balance of temperature and pressure difference. As the diameter increases, the balance of the flow is hurt which means that the cells near the front of the battery see a greater amount of flow than near the back. This in turn creates a temperature unbalance that hurts the overall battery life and performance. However, as the channel diameter decreases, the pump power required to provide the adequate flow to maintain the maximum temperature of 40 C increases. Table 1 shows all the temperature ranges for all the cooling systems for our baseline. During the optimization, the effects on the other cooling system are little to none, so we will not go into any depth on the systems other than the Battery Cooling.

System	Tmin C	Tmax C
Battery Cooling	35.9	38.2
Transaxle Cooling	75.4	86.9
Engine Cooling	76.7	89.7
Electronics Cooling	52.0	68.3

Tbl.1 Temperature Range of the Coolants of the 4 Systems

Although the software is capable of optimizing several variables simultaneously, just choosing one critical component helps show the capabilities for the purpose of this paper. Because we have a known maximum temperature requirement of 40 C for the Battery Cells, the flow valve will ensure that enough flow is sent to the Battery Chiller to meet the temperature requirement. This means that we cannot look at the minimum and maximum temperatures for choosing our optimum sizing, instead we will solely focus on the temperature difference within the battery cell, and between the hottest and coldest cell. We decided to look at a range of 1 to 3 mm in 0.25 mm increments. In table 2, we can see how the temperature range was affected by the changes.

Diameter (mm)	DT in Cell (C)	DT from 1st to Last (C)
1.00	2.42	1.22
1.25	2.52	1.35
1.50	2.63	1.50
1.75	2.74	1.67
2.00	2.88	1.90
2.25	3.05	2.11
2.50	3.17	2.40
2.75	3.40	2.76
3.00	3.54	3.06

Tbl.2 Temperature Change for Different Diameter Channels

The smaller the channels the more energy required to cool the battery, either through added pump power or extended use of the Battery Chiller, so we will select the largest diameter that meets the requirements of 3 degrees change. As can be seen the 2 mm is the largest diameter that meets the requirement. It is possible that further improvements could be made by modifying the flow rate, but by increasing the diameter, we have made an improvement.

This study was for the purpose of demonstration, and although much of the model was based directly on the actual Chevy Volt, the actual numbers may not line up with what is seen because of assumptions made to due to lack of some proprietary information.

Summary and Conclusions

The results of this study demonstrate a potential for solving large systems without having to over simplify any of your critical areas using ADFlo. Once a model like this is put together, it becomes very easy to make modifications that allow for the optimization of the system. One of the most difficult issues with liquid cooled systems is the fact that the temperature of your coolant is constantly changing. With the complete system model, you can run real transients, so that worst-case scenarios do not have to be run. Instead, real case scenarios can be run because many times the worst case is not possible at the system level. Another benefit to this type of analysis is the ability to determine "what if" scenarios such as complete loss of cooling or a partial blockage through any section of your system.

This same approach can be applied to any system involving liquid cooling, from a vehicle thermal management system to mini channels found within some of today's computer chips. For system analysis, the ability to create a variable fidelity thermal/flow model is critical to helping avoid any potential issues in the field. It also allows the elimination of some of the time consuming and expensive testing once a model has been verified.

For our model, we showed the ability to optimize the cooling fin dimension without any expensive prototyping or testing for the system we ran.

References

[1] "The Chevrolet Volt Cooling/Heating Systems Explained", http://gm-volt.com/2010/12/09/the-chevrolet-volt-coolingheating-systems-explained, Dec. 2010.
[2] http://media.gm.com
[3] "Chevy Volt Specs", http://gm-volt.com/full-specifications/

[4] “Chevy Volt Battery Cooling Systems & Algorithms”, http://www.mychevroletvolt.com/chevy-volt-battery-cooling-systems-algorithms

[5] Kays, W, M. Convective Heat and Mass Transfer, McGraw Hill, 2004.

[6] Incropera, F. B., Dewitt, D. P., Fundamentals of Heat and Mass Transfer, John Wiley and Sons, 1985.

[7] Patankar, S. V., “Numerical Heat Transfer and Fluid Flow”, Hemisphere Series on Computational Methods in Mechanics and Thermal Science, 1980.