

THERMAL MANAGEMENT OF AN ELECTRICAL VEHICLE BATTERY PACKING USING 1D AND 3D CFD

CornerStone 成基應用科技



M E C H A N I C A L A N A L Y S I S

W H I T E P A P E R

www.mentor.com

With the increased concerns over the future use of fossil fuels especially in automobiles, the rise of popular sentiment towards electric and hybrid electrical vehicles (EV) is no surprise. But there is still a major area of design challenge—batteries. As the primary means for storing the energy used for electrical vehicles, the battery of choice needs to be extremely efficient yet as lightweight as possible to allow for reasonable driving distance on a single charge. Although several factors affect the efficiency of these batteries, one of the most important is temperature.

Batteries usually have a low margin for the temperatures in which they work best. If a battery is exposed to excess heat for too long, it will likely fail; on the other hand, when the battery is too cold, it is unable to operate efficiently. This poses two design challenges that are essential to overcome when looking at electrical vehicle batteries. First, the cooling pack of the battery must be designed in a way that allows excess heat to be removed quickly and efficiently from the battery cells. And second, the thermal control system for the battery must be able to warm the battery to an efficient temperature level in a reasonable amount of time for good performance. With the lack of an internal combustion engine in electrical vehicles, this becomes crucial in the parts of North America, Europe, and Asia where the temperature can reach well below freezing in winter.

With the ever shortening time constraints on auto manufacturers today, how can these system go through the numerous time consuming design iterations necessary for a best-in-class, reliable design?

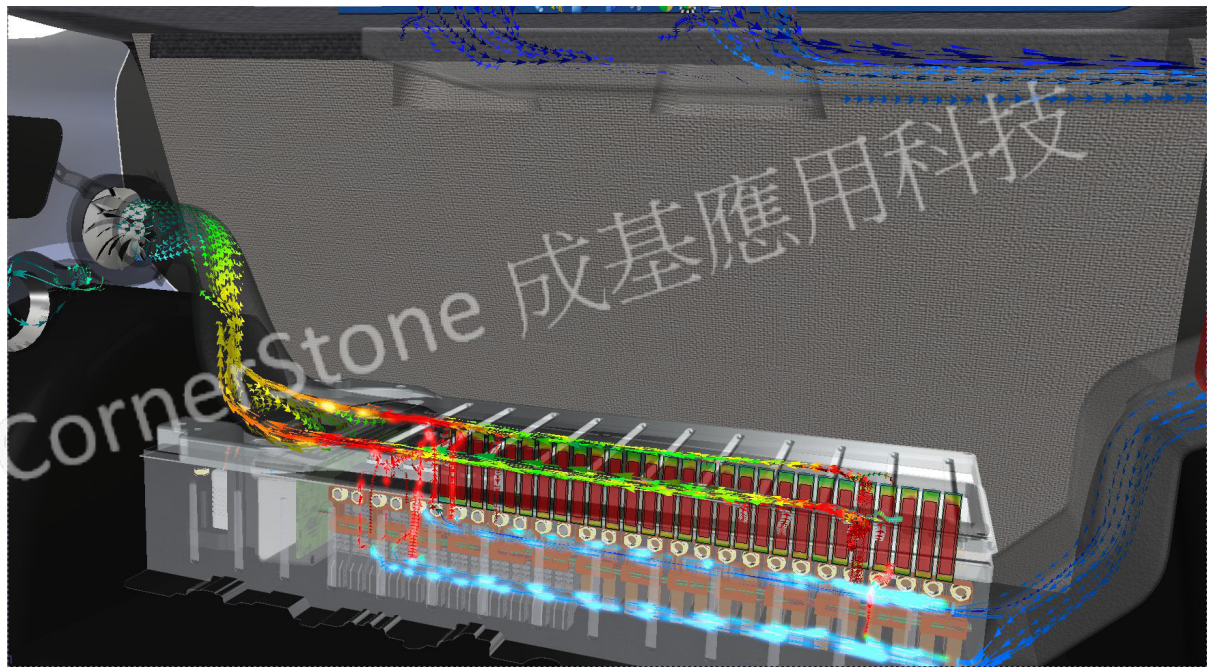


Source: Mentor Graphics FloEFD.

SIMULATING WITH 1D AND 3D COMPUTATIONAL FLUID DYNAMICS

As engineers continue to face these challenges, they need to be able to leverage all available tools to design the safest and most reliable product that is both efficient and inexpensive to produce. In the last few decades, the area of virtual prototyping has continued to grow as the suite of digital tools expanded. One of the most common engineering software tool options available is computational fluid dynamics or CFD software.

CFD is an area of fluid mechanics that uses computer-based engineering calculations to model and simulate the behavior of a liquid or gas within mechanical or electrical systems and the heat transfer throughout the model. The use of CFD to simulate the flow and heat transfer of a design means information for all variables at all locations and if necessary at every moment in time can be recorded instead of at a limited number of discrete points as with physical testing. These results can give a more complete picture of the behavior of a component or design, which gives a large scope to understand behavior that isn't quite as expected.



Source: Mentor Graphics FloEFD.

Over the years, different types of CFD software have been developed, including one-dimensional (1D) and three-dimensional (3D) options, and both of these solutions have been used successfully for modeling thermo-fluid systems in many industries, especially the automotive industry. Both 1D and 3D CFD enable engineers to improve their understanding of fluid flow and engineering designs; and in many organizations, both are used to improve product and system design and to ensure performance targets are achieved throughout the operating cycle of interest.

When engineers have access to multiple options for CFD analysis, the question arises, "When to use 1D CFD versus 3D CFD?" Although there is no definitive answer, the strengths and weakness of each approach lend themselves to two fairly defined segments.

When designing a single component or small sub-set of components, every inch of length or degree of curvature can make the difference between efficient operation and undesired fluid flow. In these cases, where small changes to a single part of a system are crucial or there are significant flow variations in multiple dimensions, 3D CFD is the obvious choice because of its ability to analyze complex geometry with extreme accuracy. However, these benefits come with drawbacks, which become more evident as the scale of the design increases. When the design reaches beyond the component level into a large system of interconnected components by pipes, ducts and hoses, the computation requirements can become too high and simulations take too long to fit within development schedules.

When this occurs, 1D CFD is a good choice. Because the 1D approach simplifies the 3D geometry to the component level, usually characterized by some sort of performance data, this type of analysis uses much less computing power and is usually significantly faster than a comparable 3D model. One of the biggest challenges with 1D CFD modeling is getting performance data that can adequately define the 3D geometry of the component at the system level. This has historically been done in one of several ways, including data from a supplier, physical testing, or empirical data from text books that is often available for standard geometries such as bends and junctions. Although these methods are adequate, it can often be time consuming to wait for a supplier to provide data; and one of the main goals of virtual prototyping is to understand the system before physical testing.

With this information, how best can 1D and 3D CFD be applied to electrical and hybrid vehicle battery design?

DESIGN CONSIDERATIONS

When using CFD, or any engineering tool for that matter, we need to understand what considerations have to be made to ensure the right tool can be used at the right time. In the example we use here, we started the design process by establishing an accurate model of the battery pack and the associated battery thermal management system (BTMS) that performed adequately under the worse-case scenario, as well as established a characterization of performance over a range of expected operating conditions. These types of analysis are best done with a 3D CFD program such as FloEFD™ so that results can account for the complex geometry and flow patterns that will occur. Once this information is available, we can focus on understanding how the battery pack interacts in the larger system, such as for a warm-up scenario on a very cold day. For this type of analysis, a 1D CFD program such as Flowmaster® is the better choice because it can leverage the characterization of the BTMS in FloEFD and analyze the interaction between the components more rapidly.

Even though both of these options deal with fluid dynamics, each has different constraints that need to be accounted for in the design and analysis of the model.

3D CFD DESIGN CONSTRAINTS

3D CFD can be a very complex solution to very challenging engineering problems. Historically, CFD has been a discipline reserved for highly trained specialists; however, recent developments from Mentor Graphics in concurrent CFD have allowed designers and engineers to analyze pressure drop and thermal performance without the need to call in the CFD specialists. Embedded into the MCAD environment, this intuitive process allows designers to optimize a product during the design stages, reducing simulation time by as much as 65 to 75% in comparison to traditional CFD tools. Because FloEFD runs natively from the same MCAD software that the battery cooling pack is designed in, there is no need to create a separate CFD model, and any redesigns that might be necessary can be done right there in the software.

The key function of the BTMS is to maintain optimum operating temperatures throughout the battery pack which ensures adequate battery performance during worse-case operating conditions. Effective thermal control is also important from a durability perspective because temperature extremes will cause the battery cells to degrade (age) faster and fail earlier. FloEFD provides results on flow paths through the battery pack as well as pressure drop, velocity, heat transfer, and local temperatures that can be used to evaluate and characterize its performance (Figure 1).

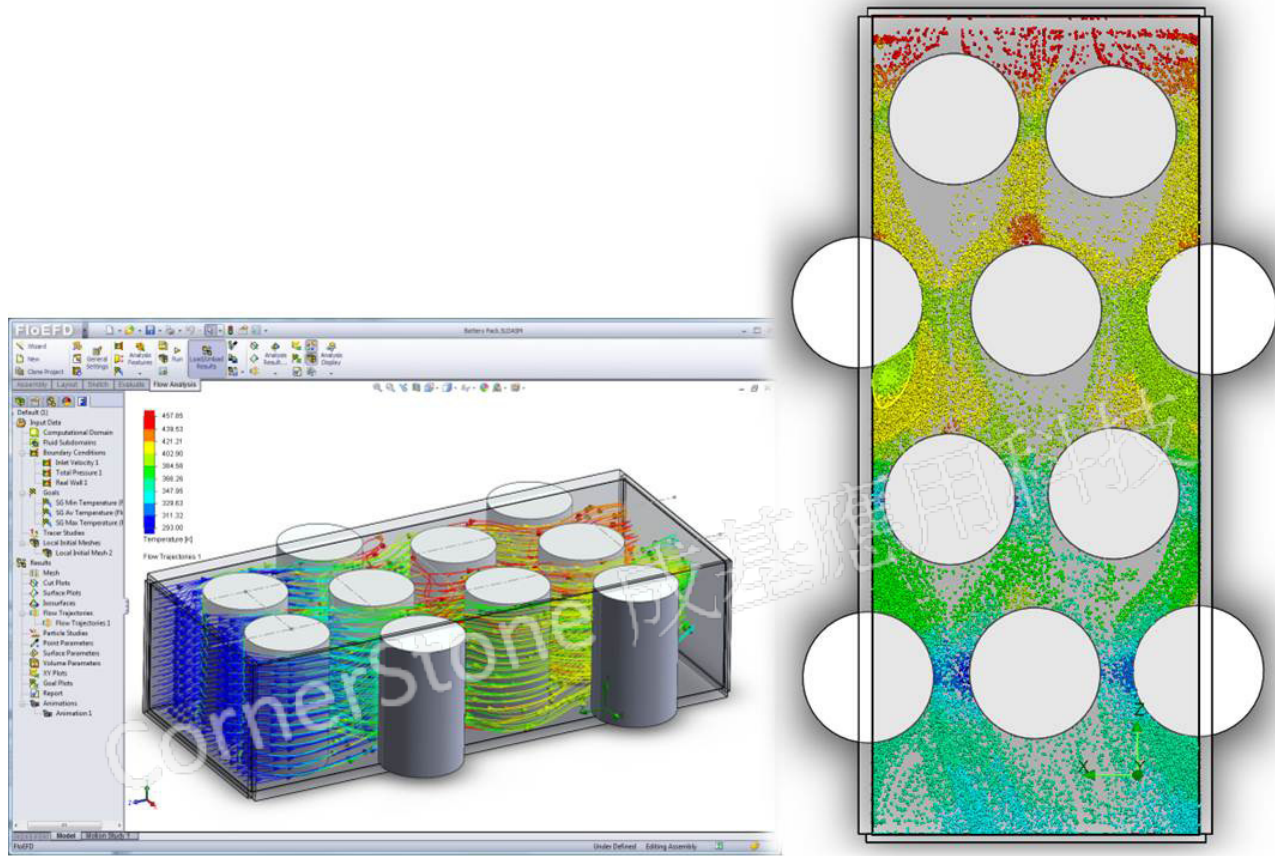


Figure 1: A typical battery pack modeled in FloEFD.

In the following example, we will consider a liquid-cooled battery pack comprised of a single layer of large cells divided into six rows with eight cells in each row, which could be considered typical of a medium size battery pack; that is, somewhere between a Leaf (300 kg) and a Tesla (450 kg). The design possibilities can vary greatly; and the design of the pack can vary frequently during the design process because it has to be optimized to achieve minimum pressure drop and uniform temperature distribution. Thus, 3D concurrent CFD provides the best solution for a faster iterative approach than using traditional CFD software and can give a good characterization of such a model.

1D CFD DESIGN CONSTRAINTS

When using a 1D CFD tool such as Flowmaster, the focus of the analysis is on the overall performance of the system and identifying how the components of the system interact, which is difficult to capture using 3D CFD. The 1D modeling approach can be used to address many of the early design

considerations such as cooling at peak conditions, warm-up during cold-start, and pump performance versus battery load.

In this example, we look at an electrical vehicle, which is a slightly less complex system than a hybrid; the liquid cooled BTMS comprises the aforementioned 6 x 8 cell battery pack coupled with a preheater, pump, and coolant reservoir. Based on the necessary power supplied to the electric motors during full acceleration, we determined that maximum heat rejection of the individual battery cells to be around 30 W per cell.

One of the important design considerations is how to represent the battery pack. It would be easy, and fully acceptable, to model it as a single lumped component using an overall heat transfer coefficient and combined thermal duty. However, in this example, the battery pack was modeled as each individual cell to ensure any temperature gradients across the pack were captured so that this information could be used if any redesigns were necessary. Also, because the pressure loss and heat transfer associated with the piping in the system were considered negligible, those parts of the system were excluded (Figure 2).

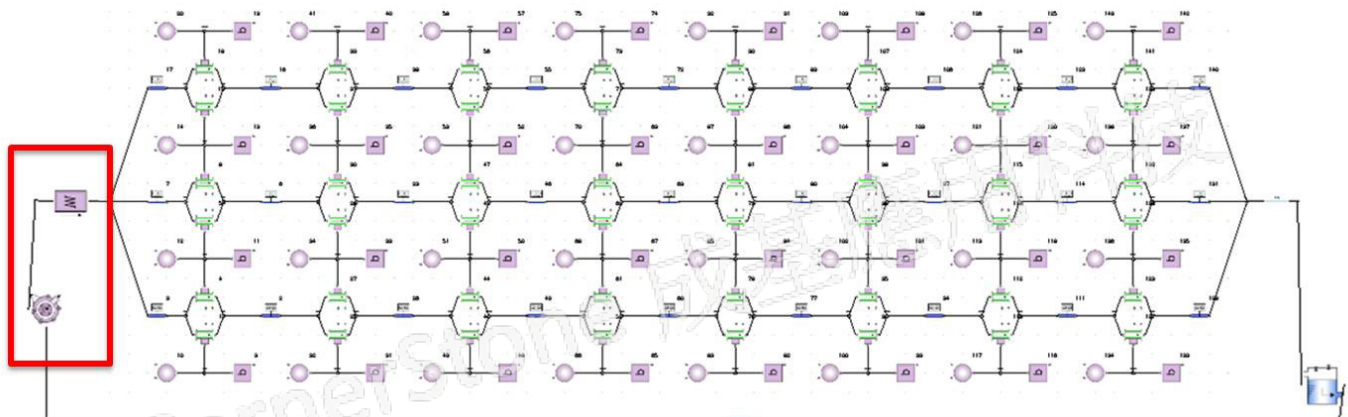


Figure 2: Individual cooling packs modeled in Flowmaster.

One of the biggest benefits of using 1D CFD is how quickly an analysis can be completed, especially for transient analysis. However, 1D CFD requires significant amounts of data to describe the physical phenomenon going on inside each component. But because Flowmaster has a substantial built-in empirical database for a wide range of geometric components such as valves, bends, and junctions, the need for finding performance data from testing or from the manufacturer is greatly minimized.

Using both 1D and 3D CFD makes it easy to use the characterization data from 3D CFD inside of the 1D model. For example, in this model, the characterization data of the pressure loss as well as the heat transfer coefficients from the FloEFD analysis were used in the battery thermal management system components in Flowmaster.

ANALYZING THE BATTERY THERMAL MANAGEMENT SYSTEM

Once both the 1D and 3D CFD models are constructed, some data can be used for analysis in both software programs; in this example, the maximum heat rejection rate.

COOLING AT PEAK CONDITIONS

The goal of the cooling at peak conditions study was to find an appropriate volumetric flow rate to keep the system under the 40 °C critical temperature for a lithium-ion battery. The 30–40 °C temperature band is the general guideline the industry has been following to preserve the lifetime of these costly battery packs. This was a worse-case study to make sure the system was able to meet the heat rejection requirements without the use of the cabin A/C circuit that manufacturers are using in very high-demand cooling applications.

We conducted a parametric study by holding the heat rejection into the cooling system at 30 W/cell and varying the pump flow rate from 2 to 15 l/min to determine the minimum pump volumetric flow rate required to keep the battery pack under 40 °C using a 50/50 glycol/water coolant and 20 °C ambient temperature.

As shown in Figure 3, the minimum flow rate that ensured all sections of the battery pack were maintained below 40 °C was approximately 9.5 l/min (2.5 gal/min). A simple crosscheck with the pump supplier ensured that the pump selected for the system met this requirement.

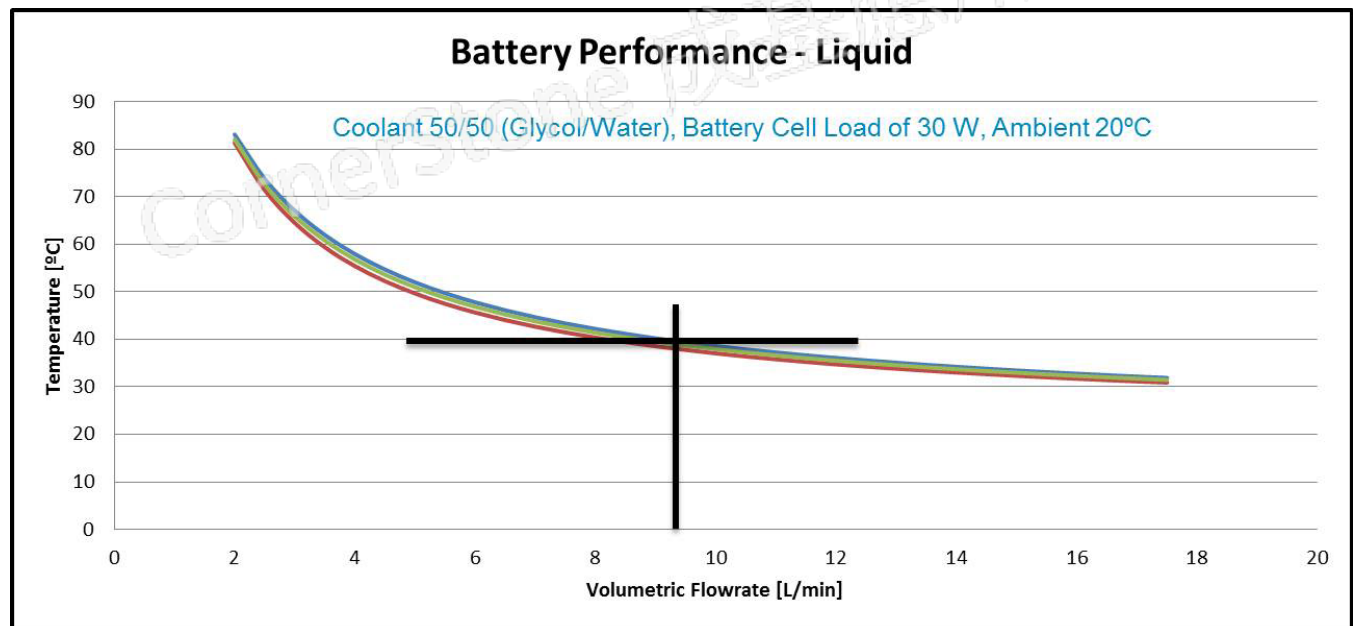


Figure 3: A minimum flow rate ensured all sections of the battery pack were held within specifications.

PUMP PERFORMANCE VERSUS BATTERY LOAD

With the minimum volumetric flow rate determined, we conducted a study to understand how the battery pack would perform under a range of operating conditions, particularly changing coolant flow rates and cell load. By again running a parametric study in FloEFD, a valuable surface map of cooling pack temperature versus coolant flow rate and cell load was created from the results (Figure 4).

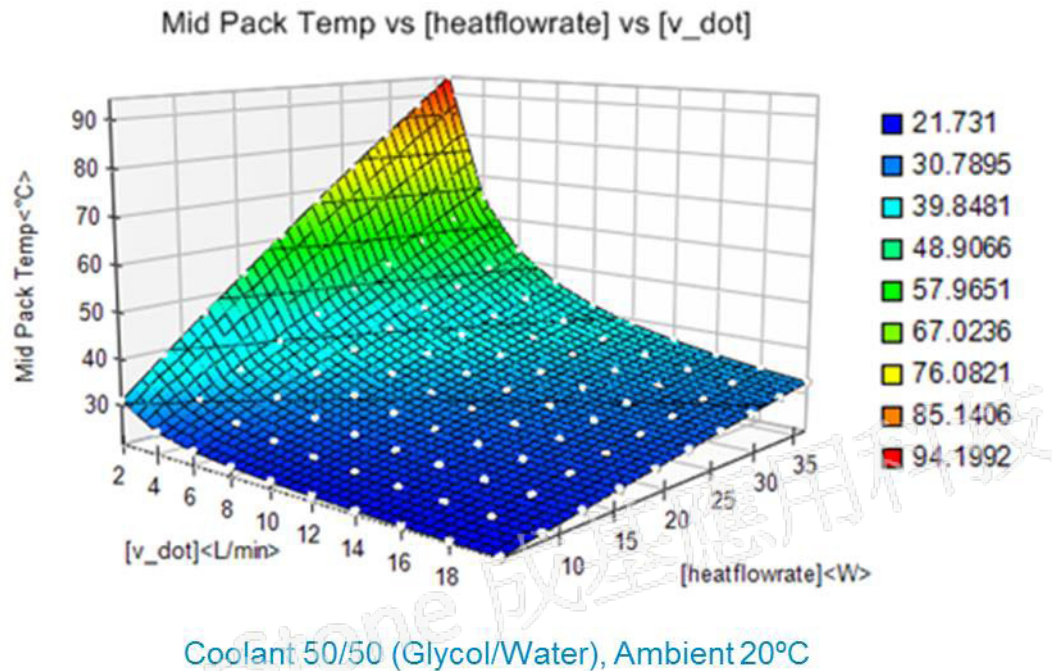


Figure 4: Surface map of cooling pack temperature versus coolant flow rate and cell load from the results of a parametric study of the cooling pack run in FloEFD.

This map provided a quick reference to the ideal conditions for which the battery pack should be exposed and presented an opportunity for redesign if it was deemed unacceptable.

From the surface map, a trendline was produced showing battery cell load versus pump performance (Figure 5).

This data was used as a starting point for tuning of the pump control algorithms. For example, an intended battery cell load of 20 W required a coolant flow rate of 5.5 L/min, which allowed the battery to stay below the critical temperature of 40 °C. In addition to creating a pump control algorithm, the surface map could be used as a characterization of the battery pack's thermal performance. Along with the pressure drop data from the study, this information was then used to model the battery pack in Flowmaster.

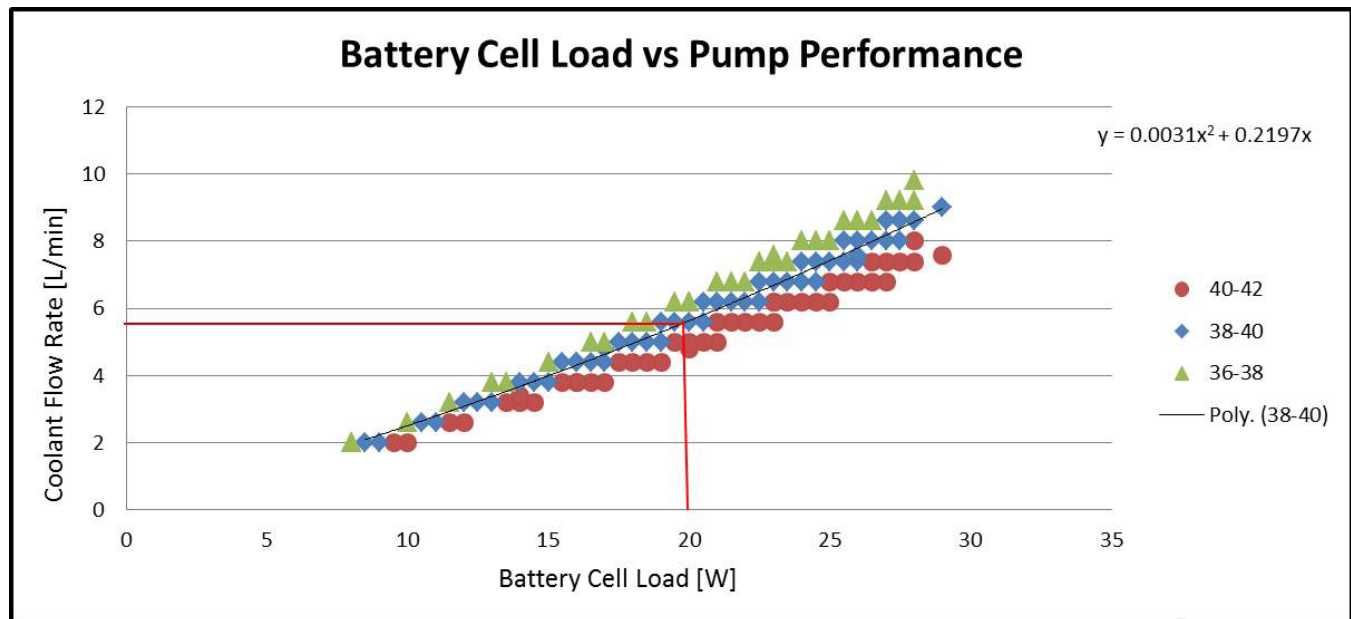


Figure 5: Trendline for sustained battery cell load with coolant flow rate. A thermal engineer could use this as a starting point for any tuning of pump control algorithms.

WARM-UP FROM COLD START

After the initial component level design has been deemed acceptable with 3D CFD analysis, it is important to investigate how it will perform within a system. A critical real-life challenge for electrical-vehicle battery systems is warming up from a cold start because there is no internal combustion engine to provide heat. Imagine it is a cold winter day in Chicago at -10 °C and the vehicle battery pack needs to reach an acceptable operation temperature (AOT) within a reasonable time.

With no internal combustion engine, a source of heat for cold startup, such as a positive temperature coefficient (PTC) heater, needs to be installed. In this simulation, two quantities could be varied—pump speed and PTC heater size—and the acceptance criteria was for the battery pack to reach 20 °C in less than 30 minutes.

An initial study was carried out with a high and low pump speed with a range of heat input of 2 kW to 10 kW. As shown in Table 1, the pump speed does not have much influence over the total time to warm up, and running it at half speed would take a lot of load off of our auxiliary battery while not affecting warm-up time.

TABLE 1: TIME (MINUTES) TO REACH MIDPACK TEMPERATURE OF 10 °C

	2 kW	3 kW	4 kW	5 kW	6 kW	7 kW	8 kW	9 kW	10 kW
500 rpm	36.5	24.2	18.1	14.5	12.1	10.3	9.1	8.1	7.3
1,000 rpm	37.0	24.5	18.3	14.6	12.2	10.4	9.1	8.1	7.3

The two quantities we wanted to change were “pump flowrate” and “heater power” (Table 1). After running the transient parametric study, we can see that, for the conditions investigated, pump speed had little influence over the total time to warm up and that running it at half speed would take significant

load off of our auxiliary battery while not affecting warm-up time. So the variable we are concerned about was system heater power.

Therefore, the design parameters could be simplified to only varying the size of the heater. As seen from Figure 6, the smallest PTC heater that would work for the system is 3 kW to reach an AOT of 20 °C in less than 30 minutes.

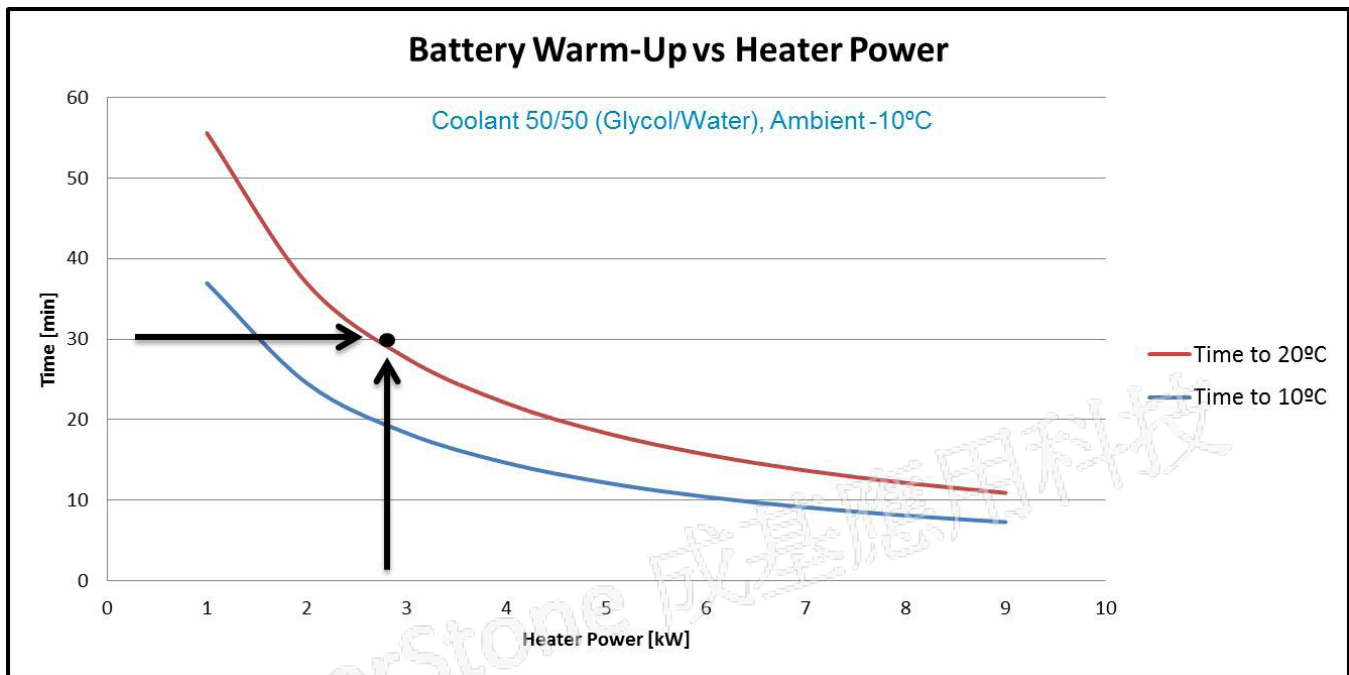


Figure 6: We needed a 3kW heater to reach an acceptable operating temperature of 20 °C in less than 30 minutes.

An interesting point to look at is the difference from when the front of the pack versus the rear of the pack gets to the AOT during battery warm-up. Figure 7 shows that the rear cells in the pack took approximately five minutes longer to reach the temperature of the front cells, which were nearest to the heater.

Another interesting data point was the difference between front and rear cells at any one point in the warm-up study. A snapshot showed a difference of 5 °C between the front and rear of the pack. If that was unacceptable for the design objectives, the study could be re-run with a higher flow rate, an extra heater added for the rear cells, or the flow geometries reconfigured to create better flow around the pack.

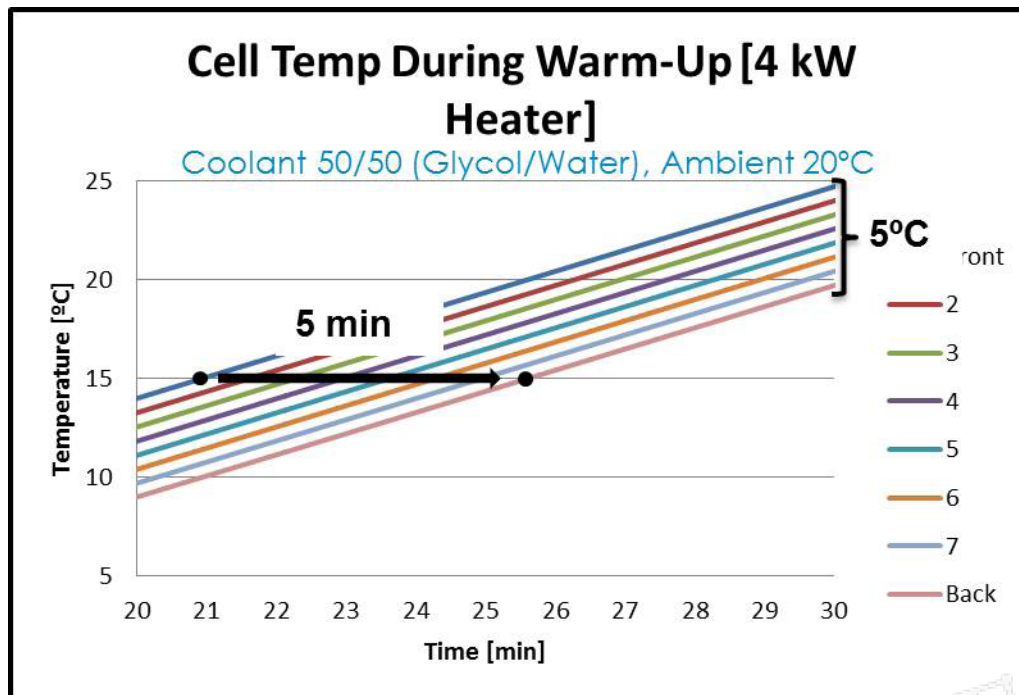


Figure 7: This graph shows that the rear cells took approximately 5 minutes longer to reach the same temperature as the front cells, which were nearest to the heater.

CONCLUSION

These examples show how advanced 1D and 3D CFD simulation techniques provided by Flowmaster and FloEFD can be used for the complex design of the thermal management system for electrical-vehicle battery packs. When concerned with analysis at the component level, FloEFD allows the study of detailed flow and thermal behavior of the thermal management system to ensure adequate performance of the battery pack. Any unacceptable operation such as misguided flow patterns or extreme thermal gradients can be identified earlier in the design process; and because FloEFD is used concurrently in the MCAD model, it can reduce simulation time by as much as 65 to 75% in comparison to traditional CFD tools. In conjunction, when the performance of the entire battery cooling system is the main concern, Flowmaster can be used to analyze how the components interact, ensuring proper system performance. With the use of both FloEFD and Flowmaster, engineers have access to the right tool at the right time for optimal design efficiency.

For the latest product information, call us or visit: www.mentor.com

©2013 Mentor Graphics Corporation, all rights reserved. This document contains information that is proprietary to Mentor Graphics Corporation and may be duplicated in whole or in part by the original recipient for internal business purposes only, provided that this entire notice appears in all copies. In accepting this document, the recipient agrees to make every reasonable effort to prevent unauthorized use of this information. All trademarks mentioned in this document are the trademarks of their respective owners.

Corporate Headquarters
Mentor Graphics Corporation
8005 S.W. Boeckman Road
Wilsonville, Oregon 97070-7777
Phone: +1 503 685 7000
Fax: +1 503 685 1204

Sales and Product Information
Phone: +1 800 547 3000
sales_info@mentor.com

Mechanical Analysis Division
(formerly Flomerics)
81 Bridge Road
Hampton Court
Surrey, UK KT8 9HH
Phone: +44 (0)20 8487 3000
Fax: +44 (0)20 8487 3001

Flowmaster Limited
(a Mentor Graphics Company)
The Maltings, Pury Hill Business Park,
Alderton Road, Towcester
Northants, UK NN12 7TB
Phone: +44 (0) 1327 306000
Fax: +44 (0) 1327 306020

Mentor
Graphics®

MGC 06-12

TECH10710