TIME DEPENDENT SIMULATION METHODS FOR VEHICLE THERMAL MANAGEMENT

Ilhan Bayraktar
Advanced Products Group
Oshkosh Corporation
Oshkosh WI 54902

ABSTRACT

Increasing power requirements along with weight and space constrains requires implementation of more intelligent thermal management systems. The design and development of such systems can only be possible with a thorough understanding of component and system level thermal loads. The present work implements 1-D and 3-D unsteady CFD based simulation tools in vehicle design process. Both under-the-hood cooling and HVAC systems are simulated in various operating conditions on a HPC Computer Cluster. System variables are optimized with gradient based BCSLIB and SciPy optimization libraries. The simulation results are compared and validated with experimental tests.

INTRODUCTION

Military ground vehicles are designed to operate in a wide range of system loads and environmental conditions. Vehicle systems and sub-systems needs to perform adequately under both extreme hot, humid and cold conditions with variable loads on engine, generator, drives, battery pack and HVAC system. Fuel efficiency and crew comfort are also affected by the severe operational and environmental conditions. Modern vehicle thermal management system has to effectively operate to ensure high availability of vehicle systems and sub-systems.

Thermal management system design considerations for under-the-hood cooling have changed significantly in recent years. In addition to the adequate cooling capability, modern under-the-hood cooling systems have to be designed to increase fuel efficiency, engine performance and life. They also need to be low cost, light weight and effectively packaged. Electric pump and variable rate impeller applications, improvements in magnetic and viscous clutches, new split cooling system designs are all aim at improving the performance of the thermal management system. New coolant management strategies can also speed-up the engine warm-up and reduce vehicle CO_2 emission.

Recent developments in HVAC systems have been focused on new refrigerant technologies and component efficiency improvements. New micro channel and foam heat exchanger concepts, developments in R-134a replacement refrigerants like R744 and HFO-1234yf and new computational design tools to streamline the system design process show the potential of next generation highly efficient, cheap, light weight and environmentally friendly HVAC systems.

Under the influence of aforementioned developments in the vehicle thermal management systems, the present work shows a heavy hybrid diesel electric vehicle thermal management system with multiple under-the-hood and HVAC loops. Both under-the-hood and HVAC thermal management systems are simulated using transient 1-D and 3-D state-of-the art computational simulation tools.

UNDER-THE-HOOD COOLING

Most of the energy in the fuel is converted to heat before it is transferred to the wheels. The largest energy loss occurs at the engine level during the combustion process. The thermal energy generated by the combustion process or by the mechanical friction needs to be managed to ensure proper system performance and long vehicle life.

Modern ground vehicle under-the-hood cooling systems do not only cool the engine, but also manage the component temperatures of generator, drives, auxiliary power units and battery packs. A good cooling system also ensures quick engine warm-up and reduce environmental pollution.

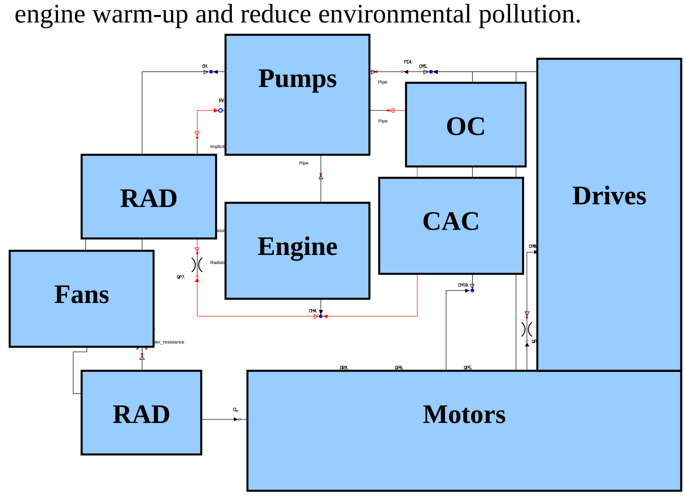


Figure 1. 1-D under-the-hood cooling model showing a two-loop cooling system for a hybrid diesel-electric truck.

The present work investigates the cooling system performance in two extreme environments, 60% tractive effort up to 130^0 F ambient conditions and engine warm-up event at -25^0 F ambient conditions [1-3]. The component temperatures are calculated with 1-D and 3-D simulation tools. Figure 1 shows the 1-D cooling loop model generated in Easy-5 software for a heavy hybrid diesel electric vehicle. During the cooling system evaluation, three different package configurations are tested. Figure 2 shows the cooling packages considered in the present work, and Table 1 shows the temperature values calculated for main powertrain components at 130^0 F ambient condition during a 60% tractive effort. It should be noted that the reported temperatures present the converged values at 130^0 F ambient condition. The results show that Design 1 and 2 are not feasible because of high temperature and packaging issues.

Design 1 *Design 2* *Design 3*

Figure 2. Three different cooling packages considered in the vehicle design.

Table 1. Converged component temperatures during 60% tractive effort simulation at 130 F ambient.

Component	Design 1 [F]	Design 2 [F]	Design 3 [F]
Pump	195.5	212.6	192.4
Radiator	175.2	194.9	174.6
Drives	178.2	197.8	177.5
Motors	183.2	202.9	182.7
Power Elec.	186.3	206.6	185.5
Generator	191.6	211.6	190.0
CAC	197.7	204.6	184.5
Oil Cooler	201.1	214.9	195.1

The cooling system performance is also affected by the airflow and convection/radiation between the under-the-hood components. 1-D design tools have limited ability to predict the influence of airflow and component layout on the system. Therefore condenser, radiator, transmission cooler and charge-air-cooler placement and their interaction with the fan/shroud combination needs to be analyzed in a 3-D environment to ensure minimum recirculation and maximum airflow and system performance. Tip clearance, immersion and penetration optimization for shroud/fan combination also needs to be performed in 3-D CFD environment. Figure 3 shows a 3-D CFD model for such under-the-hood cooling system simulation. Most commercial CFD software programs are capable of calculating cooling system component temperatures using a conjugate heat transfer model [4,5]. The Figure 3 presents the increase in the air temperature while it passes through the cooling system. The present study uses such 3-D CFD approach coupled with 1-D Easy5™ model to calculate air side restriction and recirculation parameters.

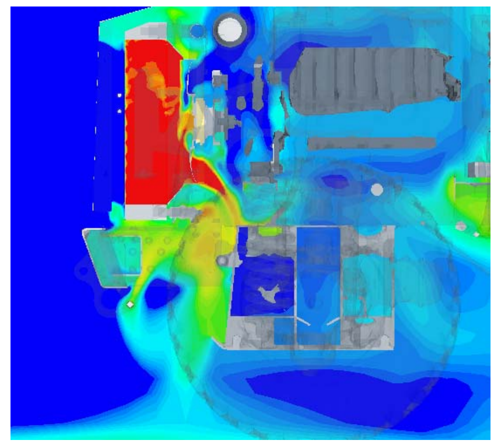
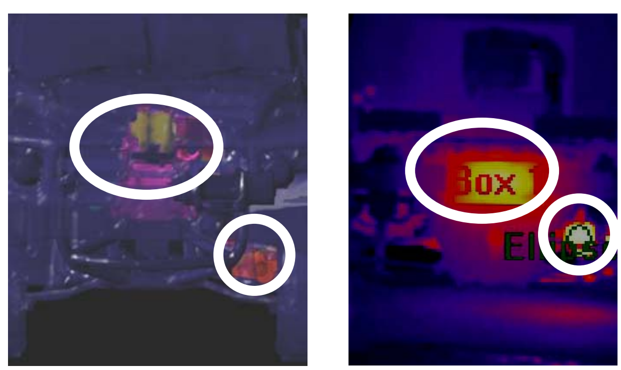


Figure 3.Temperature distribution on the longitudinal cross sectional plane.

In addition to the performance, the vehicle thermal signature is also considered during the cooling system design for military vehicles. Surface temperatures, air inlet/outlet locations and their shapes need to be analyzed to minimize the thermal signature of the vehicle. Figure 4 compares the temperature distribution between a 3-D under-the-hood model and field measurements.



(a) (b)

Figure 4. Surface temperature simulation (a) and field measurement (b) comparison.

COMPUTATIONAL TOOLS

Time dependent computations, especially 3-D CFD models, require large-size computing capability. The thermal management calculations presented in the present work are computed on an in-house computer cluster. The computational cases are managed by SGE™ (Sun Grid Engine) queue management software and Rocks Cluster Toolkit™ [6].

The parallel CFD computations are distributed on multiple CPUs using METIS™ partitioning algorithm and HP MPI™ interface. The job submission script are generated with Shell scripting and Python languages. NumPy and SciPy libraries along with Design of Experiment (DoE) tools are utilized on optimization studies [7,8].

CABIN THERMAL MANAGEMENT

Heating, cooling and ventilation requirements for most military ground vehicles are specified in MIL-STD-1472 and NATO AECTP 200 [1-3]. In general the cooling requirements in these standards pose greater challenge than heating requirements in HVAC design. Therefore most of the attention under this topic is given to the cabin cooling prediction.

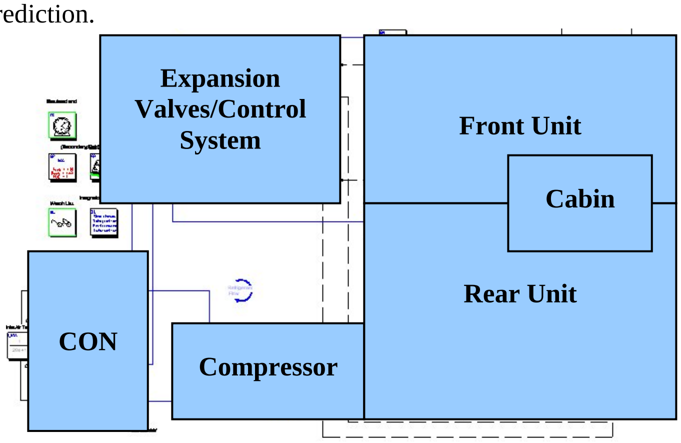


Figure 5. 1-D air conditioning system loop with two evaporator units.

A simple 1-D AC cooling loop model is presented in Figure 5. The model contains two parallel condensers and two evaporator units. The TXV valves are modeled with a simple control algorithm to ensure proper sub-cool and super-heat temperature values. The condenser and evaporator performance data is imported from the vendor-supplied test data. The cabin component in the model accounts for volume, surface area and heat transfer coefficients for simplified heat loads and the vehicle exterior surfaces. However, it was observed that the uncertainty in the thermal variables in the cabin component greatly influence the transient response of the cabin model.

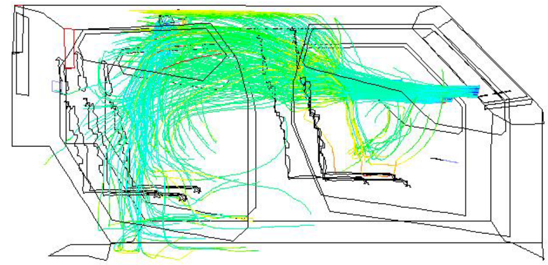


Figure 6. Air flow streamlines inside the cabin.

One approach to predict the cabin temperatures in the transient environment is to calculate the air flow and heat transfer inside the cabin using 3-D CFD tools. A simplified 3-D CFD model of a cabin showing air flow streamlines are presented in Figure 6. After the air flow information is obtained using the Fluent™ software, the results are imported into RadTherm™ package to simulate the transient thermal response of the cabin. The temperature distribution of internal surfaces after 30 minutes is presented in the Figure 7.

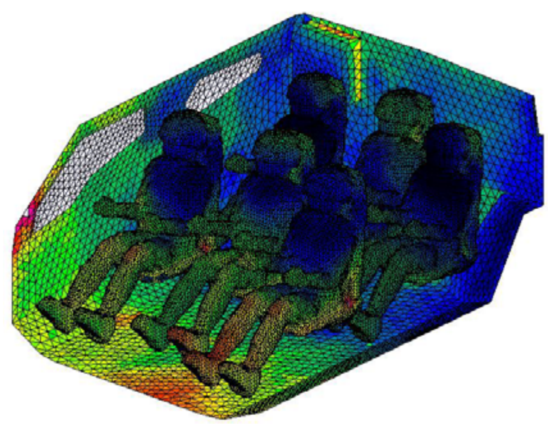


Figure 7. Surface temperature variation inside the cabin after 30-minutes.

Another approach to obtain the transient thermal behavior of the cabin is to test it in a climate chamber. The Figure 8 presents the average cabin temperature during a 60-minute pull-down test. The vehicle for this test is instrumented with more than 100 thermocouples and many pressure transducers on both air side and refrigerant side. The average cabin temperature is obtained from the occupant temperatures, which were measured based on the SAE J1503 specs.

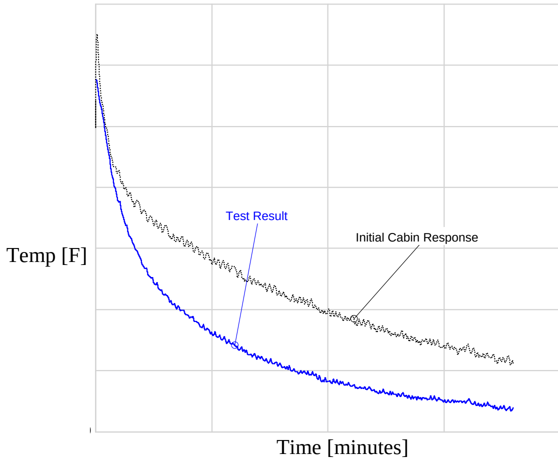


Figure 8. Average cabin temperature during 60-minute pull-down test at 130 F ambient.

Optimization

After obtaining 3-D CFD simulation results and climate chamber test numbers, the cabin component in the 1-D AC model can be optimized with common optimization tools such as gradient based BCSLIB and SciPy optimization libraries. The selected variables for this particular optimization study are Cabin Air/Wall UA (v1), Cabin/ambient UA (v2), Cabin solar heat load (v3) and Cabin air/Equipment UA (v4). The variables and their limits are set before the starting the optimization procedure. The optimization target was specified to minimize the difference between the experimental and computational results at four checkpoint locations (t=10, 20, 40 and 50 mins).

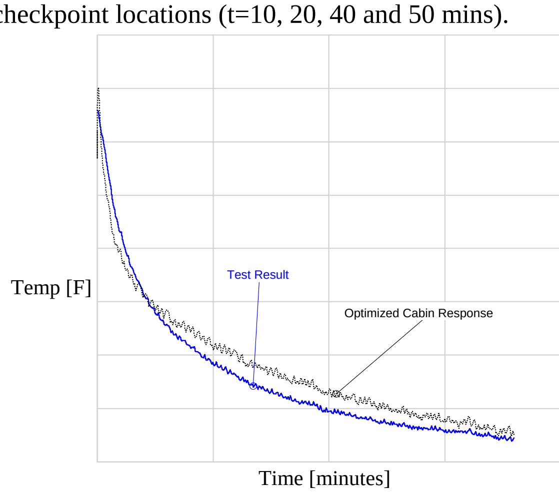


Figure 9.Comparison between the test results and optimized cabin component in 1-D model.

After the cabin component optimization the 1-D model response got much closer to the test results (Figure 9). The other component variables in the 1-D model are also compared with the test results and optimized accordingly. It needs to be noted that optimization algorithms can be converged into various different solution, and it is necessary to "guide" the optimization solver with appropriate variable limits.

CONCLUSION

Under-the-hood and HVAC systems for a military vehicle are analyzed for various environmental conditions and system loads using 1-D and 3-D CFD simulation tools. Although 1-D simulation tools provide valuable first prediction about a proposed system, it is observed that analyzing transient behavior of a thermal system requires input from higher fidelity 3-D CFD computations and sometimes climate tests. Using such input straightens the accuracy of the computational modeling and create valuable database for future vehicle developments.

REFERENCES

[1] MIL-STD-1472F, "Department of Defense Design Criteria Standard - Human Engineering", Aug. 1999.

[2] NATO AECTP 200, "Environmental Conditions" Jan. 2006.

[3] SAE J1503, "Surface Vehicle Standard- Performance Test for Air-Conditioned, Heated, and Ventilated Off-Road, Self-Propelled Work Machines", Sep. 2004.

[4] Bayraktar, I, Bayraktar, T. "Guidelines for CFD Simulations of Ground Vehicle Aerodynamics", SAE Paper 2006-01-3544, Oct. 2006.

[5] Bayraktar, I. et. al.. "An Assessment of Drag Reduction Devices for Heavy Trucks Using Design of Experiments and Computational Fluid Dynamics", SAE Paper 2005-01-3526, Nov. 2005.

[6] RocksClusters.org, "Rocks 5.1 Cluster Users Guide", 2008.

[7] MSC Software Corp., "Easy5™ 2008r1 User Manual", 2008.

[8] Ansys Inc., "Ansys v12 User Manuals", 2009.