

Full Vehicle Thermal Prediction by Identification Approach from Test Results

Takashi Takiguchi, Yusuke Yano, Yasuhiro Takii, and Nobuyuki Ohta
Honda R&D Co., Ltd.

ABSTRACT

With demands for enhanced environmental performance such as fuel economy, the tendency has been to reduce the amount of wind introduced to the engine room to reduce drag. Meanwhile, exhaust gas temperatures are increasing in order to reduce emissions concentrations. As a result, the temperature environments for parts inside the engine room and underfloor parts are becoming harsher, and accurately understanding the temperature environments of parts is crucial in determining Engine room component layout during vehicle development and applying effective thermal countermeasures.

Computational fluid dynamics (CFD) are effective for understanding complex phenomena such as heat generation and cooling. However, this paper reports the development of a method for accurately calculating the vehicle temperature distribution through identification from test results.

CITATION: Takiguchi, T., Yano, Y., Takii, Y., and Ohta, N., "Full Vehicle Thermal Prediction by Identification Approach from Test Results," *SAE Int. J. Mater. Manf.* 8(2):2015, doi:10.4271/2015-01-0441.

INTRODUCTION

Front grille openings are shrinking and aerodynamic covers are being added under the engine and to the underfloor to enhance aerodynamic performance, and there are also demands to reduce the size of engine rooms to help provide interior comfort. This is resulting in harsher temperature environments for heat-receiving parts such as mounts and sensors. As the pace of development increases, CAE is becoming more important in order to predict performance, and application is expanding from prediction in the single function range to the multiple function range. In particular, the scale of fluid analysis (CFD) is expanding rapidly with increased computer capabilities, and thermal fluid analysis is being performed on a full vehicle scale. Hong et al. calculated parts temperatures by discretizing airflow and heat transfer, and co-simulating two solvers [1]. In this case, however, mesh creation also needs to be performed twice, which requires a great deal of process time. Tanaka et al. obtained parts temperatures in soak mode by dividing the process into analysis of the radiator heat balance and a process that obtains the exhaust manifold surface temperature distribution [2]. However, the accuracy of the parts temperature prediction deviated from actual measurements by 20°C. It is a challenge to use CFD to predict heat damage, and in particular to accurately predict body part temperatures, so temperatures are generally verified by vehicle tests. This paper reports a method of obtaining identification coefficients from vehicle test results and applying these coefficients to a CFD model. This method is accurate enough to enable the use of CAE to determine the performance of prototype vehicles and subsequent lots.

FRAMEWORK

Figure 1 shows the process for determining the temperature of rubber, plastic and other parts. The required driveshaft horsepower is obtained from the vehicle specifications and the driving mode, and the required engine torque is the torque after taking into account the friction of elements such as the transmission and air conditioning. The engine operating point that meets the required torque is determined, and then the cooling loss and exhaust loss are fixed. The airflow around the vehicle is obtained using CFD. The cooling loss is applied to the radiator block, making it possible to predict the water temperature and radiator exhaust wind temperature from the wind velocity passing through the radiator. The exhaust loss is applied to the exhaust manifold, and the inside of the exhaust pipe is calculated using CFD. The exhaust pipe surface temperature is obtained by co-simulating the radiator exhaust wind temperature and the CFD of the inside of the exhaust pipe.

Prediction technology was created for two driving modes. The first is the key-off soak mode after reaching temperature equilibrium while climbing a hill at 80 km/h. The second is the parking idle mode after reaching temperature equilibrium while driving at a high speed of 120 km/h. The peak parts temperature values were obtained in both modes. A method was developed of extracting identification coefficients from vehicle test results and using these coefficients to increase the accuracy of vehicle parts temperature prediction using CFD. There are four identification coefficients: radiator PQ curve, exhaust pulsation conventional amplification factor (CAF), thermal conductivity, and surface emissivity. CFD temperature prediction utilizing these coefficients provides parts temperature prediction accuracy equal to that of vehicle tests.

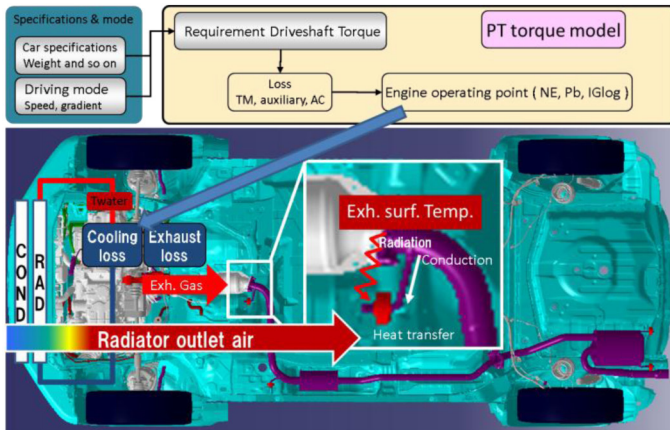


Figure 1. Process for obtaining parts temperatures from heat generation and cooling balance

CFD MODELING METHOD

The process for obtaining parts temperatures is described below.

STEP 1 Calculation of airflow

Apply the inlet velocity, heat exchanger PQ curve, and fan speed (rpm) as the full vehicle airflow, and obtain the wind velocity distribution around the full vehicle.

STEP 2 Calculation of engine cooling

Apply the heat radiation characteristics and cooling loss to the radiator, and co-simulate with the wind velocity obtained in STEP 1 to obtain the radiator water temperature and radiator exhaust wind temperature. If the water temperature diverges from actual measurements, apply a coefficient to the radiator PQ curve in STEP 1 to adjust the water temperature.

STEP 3 Calculation of catalyst surface temperature

Perform CFD for the internal flow from the exhaust manifold to the catalyst outlet. Perform unsteady state calculations using the mass flow rate and exhaust gas temperature at the engine outlet as the input conditions. Co-simulate with the wind temperature obtained in STEP 2, and obtain the catalyst surface temperature and catalyst outlet gas temperature.

STEP 4 Calculation of exhaust pipe surface temperature

Perform CFD using steady state calculations for the internal flow from downstream of the catalyst to the tailpipe. Take into account exhaust gas pulsation, and apply a CAF to the thermal conductivity of the steady flow over the inner wall of the exhaust pipe to bias to match the measured exhaust pipe surface temperature and exhaust gas temperature.

STEP 5 Calculation of parts temperatures

Perform sensitivity analysis using the thermal conductivity and surface emissivity of each material as parameters, and create parts temperature response surfaces. Obtain the thermal conductivity and surface emissivity from the point on the response surface where the part temperature is close to the actual measurement.

The detailed procedure for each STEP is described below.

Step 1 Calculation of Airflow

The subject of calculations in this study was the vehicle design model for a mass production passenger vehicle as shown in Figure 2. This vehicle has a condenser for air conditioner cooling and a radiator and fan for engine cooling. The calculation space was a rectangular space as shown in Figure 3. The calculation mesh was based on an orthogonal mesh as shown in Figure 4. Polyhedral elements were placed near the vehicle body to increase the shape reproducibility and resolve the boundary layer flows.

Table 1 shows the boundary conditions. Constant velocity and temperature conditions were used at the inlet boundary of the calculation space, and constant pressure conditions were used at the outlet boundary. The pressure loss in the radiator and condenser regions was simulated using the pressure loss characteristics relative to the through wind velocity measured beforehand in the standalone state. The fan rotations were simulated using MRF. Peripheral velocity corresponding to the inflow velocity was applied to the tire surface to simulate rotary motion.

Table 2 shows the computation method. The fluid analysis in this study consisted of numerical analysis performed using the general-purpose thermal fluid analysis software FLUENT [8]. The governing equation was a steady state 3D incompressible Navier-Stokes equation, and discretization was performed using a cell-centered finite volume method on an unstructured mesh. However, the following equation was used to simulate changes in density due to changes in temperature.

$$\rho = \frac{M \cdot P_{ref}}{R \cdot T} \quad (1)$$

Here, ρ [kg/m³] is the fluid density, T [degrees C] is the fluid temperature, R is the gas constant, M is the molecular weight of the air, and P_{ref} is the reference pressure. The turbulence model used the Realizable k- ϵ model adding revision of the viscosity coefficient to the standard k- ϵ model, and an enhanced wall treatment method that can also support low Reynold's number flows was used as the wall function near the vehicle body. The space discretization method first checked the convergence with a low-order solution, and then used a 2nd-order upwind scheme. The radiation model used the discrete-ordinate method.

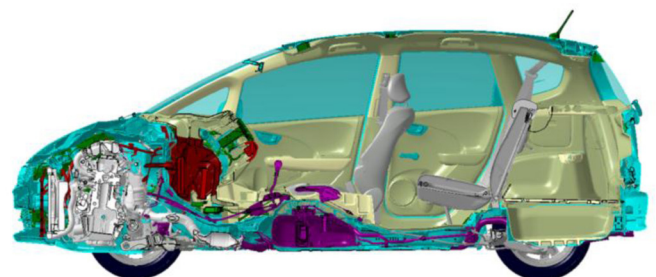


Figure 2. Calculation subject model

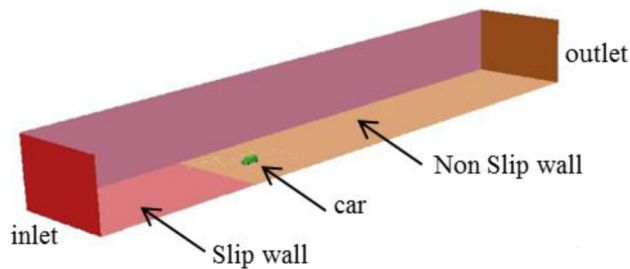


Figure 3. Calculation space

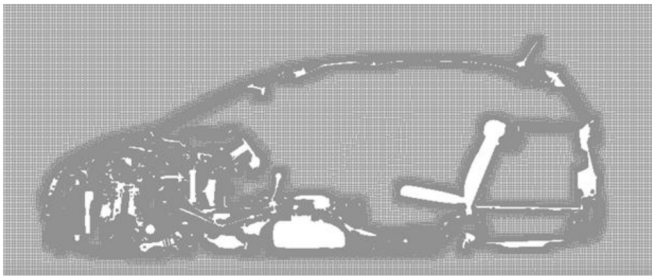


Figure 4. Calculation mesh of Full Vehicle

Table 1. Boundary conditions

Boundary	Condition
Inlet	velocity
Outlet	static pressure
Slip wall	slip wall
Non Slip wall	non-slip wall
Car	non-slip wall

Table 2. Computational methods

Governing Equation	Incompressible Navier-Stokes Equation
Turbulence Model	Realizable k- ϵ
Wall Treatment	Enhanced Wall Treatment Method
Equation Discretization Method	Cell-centered Finite Volume Method
Space Discretization Method	2 nd -Order Upwind Scheme
Time Discretization Method	1 st -Order Backward Scheme
Reconstruction Method	Least-Squares Method
Limiter Function	Slope Limiter by Barth-Jespersen
Velocity-Pressure Coupling	Pressure-Based Coupled Method
Radiation Model	Discrete-Ordinate Method

Step 2 Calculation of Engine Cooling

Estimation of Cooling Loss

1. Creation of a model for heat flow to the cooling water based on engine bench test results

To predict the radiator exhaust wind temperature, it is necessary to predict the amount of heat radiated by the radiator and the cooling water temperature in accordance with the driving state.

A method was created for increasing the accuracy of prediction of the amount of heat in response to driving conditions when mounted in a vehicle. This was done by using an experimental model based on the results of heat flow measurements during bench tests as the method of predicting heat flow from the engine to the cooling water. Figure 5 shows the created model of heat flow to the cooling water.

Regarding heat flow to the cooling water, the difference between bench tests and vehicle installation is that the amount of engine surface air cooling, alternator load, air conditioning load and other factors change according to the driving conditions, so these effects need to be properly taken into account.

As the bench test conditions for achieving this, the wind velocity in the test chamber was kept constant to maintain constant thermal conductivity on the engine surface, and the work performed by the alternator and the air conditioning compressor consisted of operation at only a mechanical load. Thus, the test was performed under conditions that made it possible to theoretically take into account the amount of cooling air and auxiliary load corresponding to the driving conditions when mounted in a vehicle.

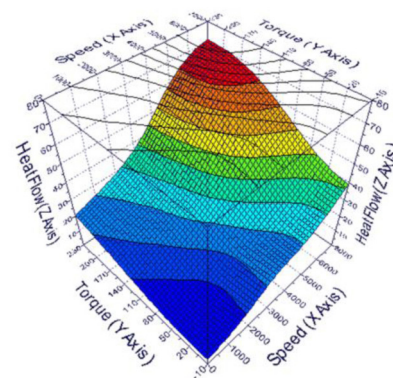


Figure 5. Heat flow to the cooling water model

2. Identification by vehicle tests and enhancement of model accuracy

The engine operating conditions can be understood from the vehicle test results, so calculation of the amount of heat from the engine-generated torque taking into account the auxiliary load was added to the previously described calculation model for heat flow to the cooling water, and the amount of cooling air was identified from the radiator exhaust wind temperature and the thermal conductivity obtained by CFD. This made it possible to create an accurate heat flow calculation model.

Figure 6 shows heat flow to the cooling water in operating point. The drivetrain transmission efficiency, intake air temperature and amount of ignition correction can be understood from advance vehicle tests, and changes in the engine speed and drive force corresponding to vehicle changes in subsequent lots can be accurately predicted by the identified heat flow model. This made it possible to increase the accuracy of prediction

of the amount of heat radiated by the radiator. In addition, the identified heat flow model can be reused in vehicle cooling system design (cooling device settings, vehicle opening settings) in the initial development stages when developing the next generation vehicle.

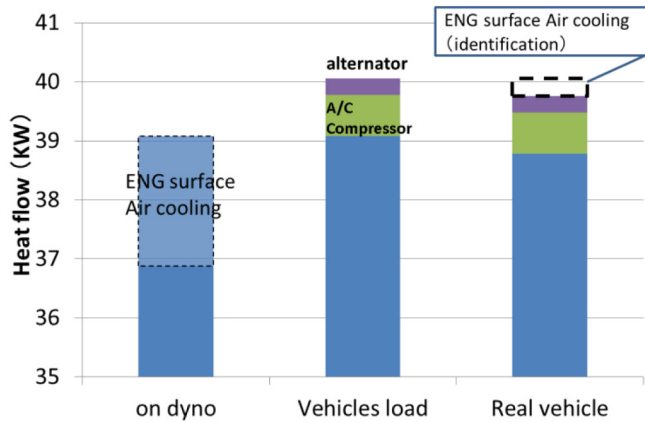


Figure 6. Heat flow to the cooling water in operating point

Calculation of Full Vehicle Cooling

Heat exchange was examined when a forced convection heat transfer environment emerges in an on-board radiator such as shown in Figure 7. Here, high temperature coolant flows inside the radiator tubes and exchanges heat with low temperature air passing outside the radiator tubes. In this study, the high temperature coolant was assumed to flow in only one direction, vertically from top to bottom, inside the radiator. Labeling the density of air as ρ [kg/m³] and the specific heat of air as C_p [J/(kgK)], the relational equation for the change in air temperature ΔT_a [degrees C] before and after the air passes through the radiator and the energy W_{rad} [W] emitted by the radiator per unit of time is expressed as follows.

$$W_{rad} = \rho \cdot C_p \cdot \Delta T_a \cdot Q_{air} \quad (2)$$

$$\Delta T_a = T_{a_out} - T_{a_in} \quad (3)$$

Here, T_{a_out} [degrees C] is the air temperature after passage through the radiator, and Q_{air} [m³/s] is the volumetric flow rate of the air passing through the radiator. It is assumed that the air temperature before passage through the radiator T_{a_in} [degrees C] can be predicted using a numerical analysis method separately from W_{rad} . Based on Eq. (2) and Eq. (3), this equation is as follows.

$$T_{a_out} = T_{a_in} + \frac{W_{rad}}{\rho \cdot C_p \cdot Q_{air}} \quad (4)$$

Therefore, to calculate the radiator exhaust wind temperature T_{a_out} , W_{rad} and Q_{air} must be predicted.

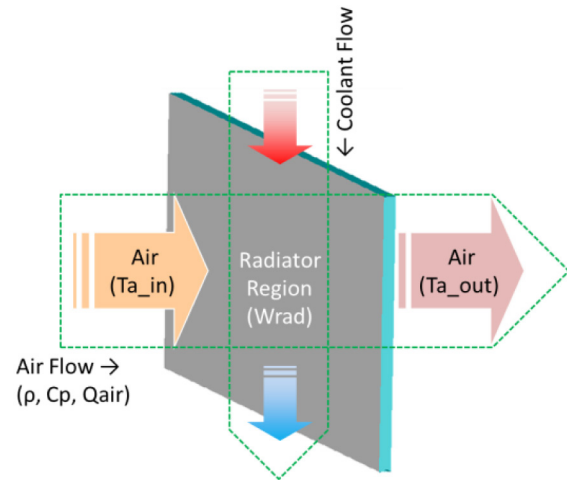


Figure 7. Outline of radiator heat exchange

Airflow Passing through the Radiator

The flow of calculations used in this study for the mass flow rate through the radiator is described below.

The amount of heat dissipation in response to driving conditions is imparted to the capacitor region in STEP 1 as a source term. The heat dissipation characteristic and cooling loss of the radiator unit are input, and the radiator water temperature and radiator exhaust wind temperature are obtained.

Here, the change in the radiator exhaust wind temperature ΔT_{a_out} when the mass flow rate through the radiator Q_{air} is changed was examined in order to investigate the allowable error of the mass flow rate calculated. Based on Eq. (4), when the mass flow rate through the radiator changes from Q_{air} to $Q_{air} + \varepsilon$, the effect on the radiator exhaust wind temperature is as follows.

$$\Delta T_{a_out} = \frac{W_{rad}}{\rho \cdot C_p} \left(\frac{1}{Q_{air}} - \frac{1}{Q_{air} + \varepsilon} \right) \quad (5)$$

Therefore, the following relationship should be met to keep ΔT_{a_out} at a given value or less.

$$\varepsilon \leq \left| \frac{\rho \cdot C_p \cdot Q_{air} \cdot \Delta T_{a_out}}{\rho \cdot C_p \cdot Q_{air} \cdot \Delta T_{a_out} - W_{rad}} \right| \quad (6)$$

In hill-climbing 80 kph mode, $W_{rad} \approx 40000$ [W], $\rho \approx 1.0$ [kg/m³], $C_p \approx 1000$ [J/(kg-degrees C)], and $Q_{air} \approx 1.0$ [m³/s]. In this case, Eq. (6) indicates that in order to predict ΔT_{a_out} within 2 [degrees C], the allowable error of Q_{air} is 5%. However, when calculations are performed using the pressure loss characteristics for the through wind velocity measured in a standalone state, the calculation results sometimes predict a mass flow rate through the radiator that is at least 20% lower than the measurement results, as shown in Figure 8.