Simulation of the Thermal Underhood Environment with CFD

It is quite narrow in the engine compartment of modern passenger cars. Additional components like intercooler, catalytic converter or climate control increase the temperatures significantly and can lead to temperature caused failures in extreme driving modes. For reliable predictions about the thermal underhood environment of Subaru vehicles in early development stages, Fuji Heavy Industries uses simulation tools like the Ansys CFD programme Fluent.

1 Introduction

The climate change has become a central topic in public discussions. And even if the individual traffic doesn't have the biggest stake in air pollution, the car manufacturers are asked to make their contribution to reduce environmental impacts. Besides the usual parts, components and aggregates, in modern cars you can find a number of solutions to increase the fuel efficiency and to reduce pollutants, CO_2 and noise emissions.

From the view point of the underhood thermal environment many of these techniques have negative effects. Noise reduction efforts employ a larger undercover, which work by sealing the underhood mechanical environment more tightly. Exhaust emissions reduction measures use additional catalysts, which raise the temperature of the exhaust pipe. Improvements of fuel efficiency lead to various heat sources like the battery due to new power systems, such as a hybrid powertrain. In all cases, the thermal environment in the engine compartment becomes more severe, besides the aspect of packaging.

2 Influence of the Development by Prediction of the Thermal Behaviour

To solve these problems, the underhood thermal environment needs to meet various conflicting requirements, as well as satisfy the heat resistance of underhood parts and components for ensuring their functionality even under extreme conditions. Iteration loops and time-consuming design changes should be minimizes, as well as the number and the extend of physical tests. Such a proceeding only can be realized, if a prediction method is available, which allows the reliable assessment of the thermal underhood conditions in an early design phase and without physical testing.

Numerous automotive manufacturers are already conducting development activities in this field. CFD-simulation programs are the core of such prediction solutions. Based on CAD-data and with appropriate constrains, fluid-thermal calculations can be made a long time before a first prototype is completed. In the following the potential, effectiveness and the limits of a solution developed by Fuji Heavy Industries (Subaru) will be shown in comparison with physical testing results [1].

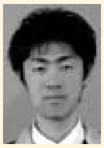
3 Physical Heat Damage Tests

To experimentally evaluate the heat balance and potential damage of components and aggregates even under severe conditions, car manufacturers run so called heat damage tests. In a closed room, temperature controller, humidity controller and solar simulator enable the setting of any value of temperature, humidity and solar radiation to reproduce various thermal environments.

The tests are performed under the high temperature conditions of summer, when the underhood heat load is highest and the air conditioner is operated in fresh-ventilation and full-cool mode. The driving modes mainly used in the tests are high-speed driving and hill-climbing, where the combustion engine is subject to a high heat load, and an idling mode, where cooling air is not delivered sufficiently.

For the results presented here, this study examined the underhood thermal environment in high-speed mode, which is although the basic mode for the evaluation of vehicle performance. **Table 1** shows the main test conditions. The test was performed with a prototype of the Subaru Impreza WRX Sti, and **Table 2** shows the specifications. Since the test vehicle is a prototype, some parts used in the underhood environment are different from those of the production model.

The Author



Eng. Takumi Hasegawa
is R&D simulation engineer in the Subaru Technical Research Center of Fuji Heavy Industries Ltd. in Ota (Japan).

Table 1: Test conditions at the heat damage test

Feature	Value	Unit
Outside air temperature	35	°C
Air humidity	70	%
Solar insolation	814	W/m²
Vehicle speed (engine speed between 3000/min and 3200/min)	33.33	m/s
Air conditioner modes	full cool, fresh ventilation on	–

Table 2: Specifications of the test vehicle Subaru Impreza WRX Sti

Feature	Description
Spark ignition engine	boxer, 2 litre displacement, double overhead camshafts, super-charging
Driveline	full-time four wheel drive
Transmission	manual shifted six-speed transmission
Dimensions of the vehicle (Length / Width / Height) in mm	4465 / 1740 / 1425

Table 3: Boundary conditions for CFD simulation

Feature	Part	Setting
Heat sources	exhaust system	temperature
	cooler	air flow resistance and heat radiation according to air flow velocity
	condensator	
	charge air cooler	
	solar simulator	temperature
Inlet	34.7 m/s at 35 °C	
Outlet	0 Pa, free emission	

4 Numerical Simulation

As already told, numerical simulation programs are the best requisite to meet the advanced process requirements in car development. By running such simulations, different design variants can be evaluated and compared in early design stages without bigger material expenses (Simulation Driven Product Development). Furthermore designers can achieve a much deeper understanding of processes and relations as it would be possible with conventional methods.

4.1 Numerical Simulation Method

For the fluid-thermal calculations the CFD-program (Computational Fluid Dynamics) Fluent 6.2 from Ansys Inc. was used. Some of the main advantages of this programme are an enormous bandwidth of physical models, its system openness und a high flexibility for the customization on a variety of different problems.

For the heat damage test described here, a steady state calculation with the mathematically realizable k-ε turbulence model and the discrete ordinates radiation mode was used. A Xeon personal computer, which had 3.4 GHz and an 8 GB memory in total, performed parallel calculation with four CPUs. The computation time was approximately 36 h for each simulation case.

4.2 Computational Model

The basic requirement for each realistic simulation is a geometrical calculation model of the test room as realistic as possible as well as a set of valid and physically reasonable constrains, **Table 3**. For the calculations, the vehicle and the test room were divided into about 230 boundaries, for which were set 14 material physical properties and ten radiation rates according to the parts. The mesh size varied from 5 to 30 mm, and the calculation model had about 5.9 million meshes in total.

Thermal boundary conditions were applied to radiator, condenser and intercooler, where heat was released according to the airflow velocity. A thermal resistance was applied to other test regions according to the heat transmission coefficient, which was calculated based on the inner structure of the components

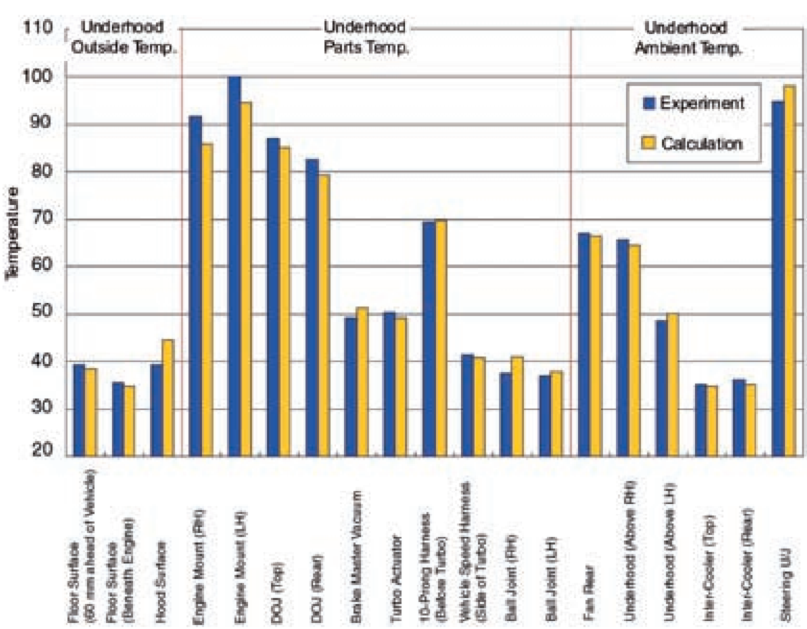


Figure 1: Comparison of component temperatures (experiment/simulation) with the air-condition off

5 Evaluation of Thermal Environment Predictability

For making a statement about the quality of the simulation results the selected driving mode was calculated with air-condition on and off and was comprised with the adequate measuring data, **Figure 1** and **Figure 2**.

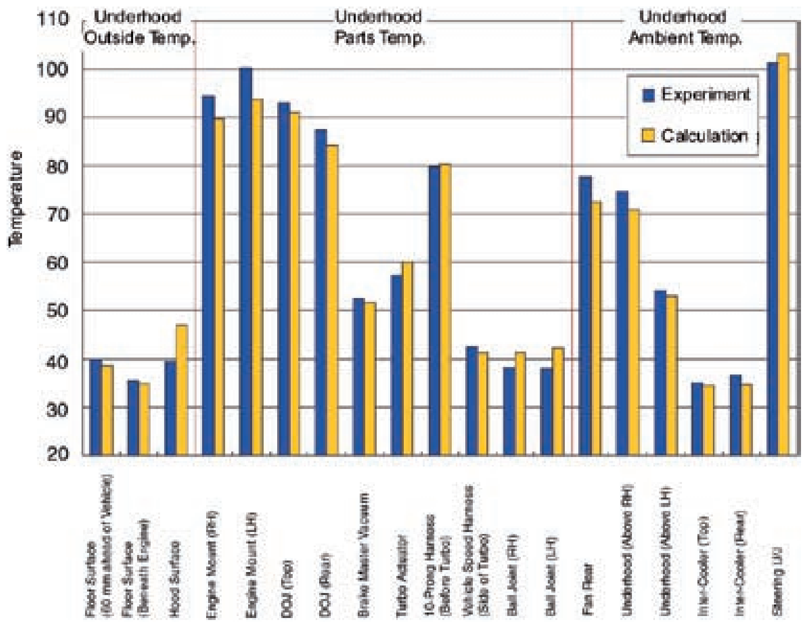


Figure 2: Comparison of component temperatures (experiment/simulation) with the air-condition on

5.1 Comparison Results

The comparison of the experimental and calculation results of the tested regions' temperatures show a sufficient compliance for both cases, air conditioner was both on and off. The temperature error was less than 10 °C.

However, closer observations indicate that the engine mount had a larger error when the air conditioner was on and off, and so do the rear of the fan and the right side of the underhood environment when the air conditioner is on. In order to improve the prediction accuracy, it was necessary to adjust settings of the heat sources more precisely.

5.2 Examination of Thermal Environment, Thermally Loaded by the Air Conditioner

In order to determine the cause of their most severe thermal environment, the above calculation results were used to visualize the parts by colour, which reach a higher temperature, for both cases, air conditioner on and off.

Figure 3 shows the airflow entering the underhood environment, and the temperature distribution in the underhood environment. It indicates that the airflow from the front grill increases in temperature as it passes through the condenser and radiator and receives heat from them. Then, the heated airflow enters the underhood environment and heats the parts. In addition, among the exhaust system parts the turbocharger reaches a very high temperature and its thermal radiation contributes to the temperature rise of the surrounding parts.

The temperature distribution was also examined for both cases (air condition off/on) for the ambient temperature of the fan's rear engine mount and double offset joint (DOJ) boot, **Figure 4** and **Figure 5**. As supposed their temperature increased when the air conditioner was on compared to when the air conditioner was off.

Like in Figure 3, these indicate that in the underhood environment the ambient temperature increased at the rear of the condenser and radiator. Especially when the air conditioner was on, the ambient temperature of the underhood environment rose sharply because the heat radiation from the radiator and condenser increased. It is conceivable that this contributed to the temperature rise at the rear of the fan.

In addition, the figures indicate that the temperature of the parts surrounding the exhaust pipe was highly subject to the radiant heat from the high-temperature exhaust system parts. When the air conditioner was on, the engine speed increased and the surface temperature of the exhaust pipe rose, which caused the surrounding parts to become heated easily. It is conceivable that this contributed to the temperature rise of the engine mount and DOJ boot, which were located near the exhaust pipe.

As can be appreciated from the above calculation results, the prediction method of this study successfully simulates

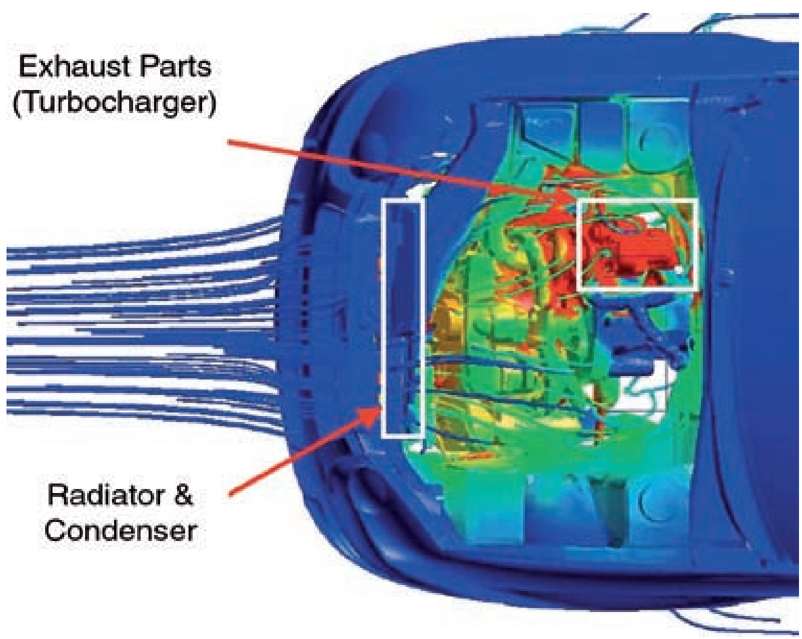


Figure 3: Visualized airflow and temperature in the underhood environment

the heat damage test for high-speed driving. In addition, the obtained temperature distribution allows identifying parts that increase in temperature. This knowledge can in turn, be applied to the study of heat damage prevention.

6 Conclusion

A prediction method for the thermal environment in the underhood environment was presented. The results from Fuji Heavy Industries (Subaru) have shown that it is possible to simulate with the CFD programme Ansys Fluent the in the heat damage test measured temperature distribution in a good qualitative and quantitative accordance.

The developed method thus gives a promising basic approach to optimize the layout of the underhood environment in early vehicle development stages. To further increase the effectiveness of the CFD programme as a prediction method in this area, the model generation and the setting of boundary conditions should be simplified without compromising the calculation accuracy.

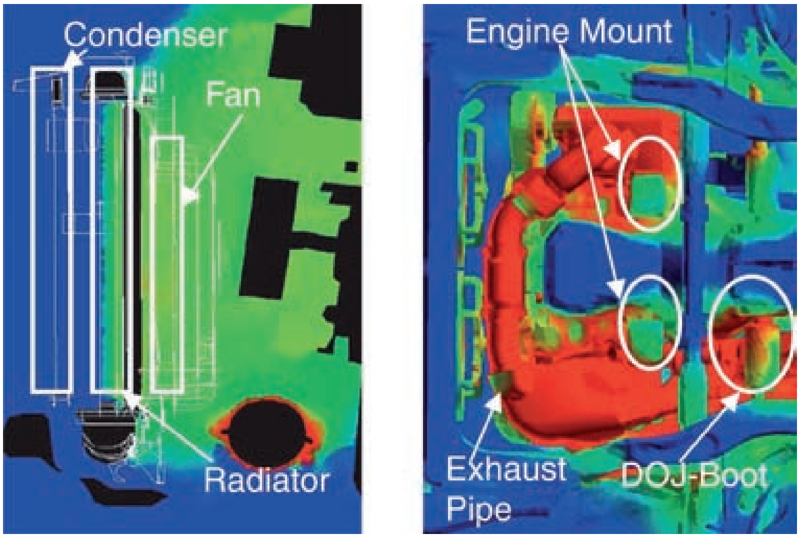


Figure 4: Temperature distribution in the underhood environment for air-condition off

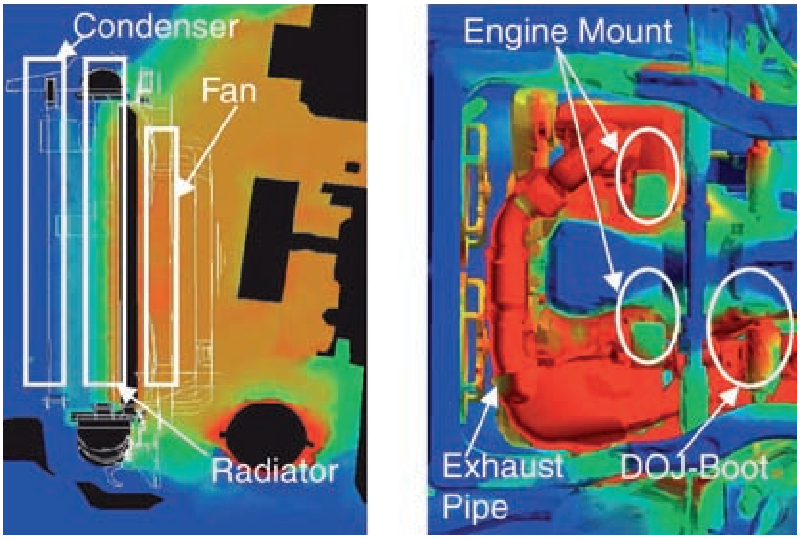


Figure 5: Temperature distribution in the underhood environment for air-condition on

Reference

[1] Hasegawa, Takumi; Komoriya, Toru: Simulation of Underhood Thermal Environment. Subaru Technical Research Center, Fuji Heavy Industries ltd., Ota, Japan; Proceedings of EACC European Automotive CFD Conference, 05-06 July 2007, Frankfurt/Main, Germany