Technical Paper

Development of Temperature Prediction Method for Components in Engine Room of Construction Machinery Using Thermal-fluid Simulation Coupled with 1D-CFD and 3D-CFD

Masaya Yuguchi

Yoshiaki Yoshida

Sho Watanabe

Takaaki Mizokami

In order to achieve front-loading of designing, it is important to identify heat spots in the engine room of construction machines and consider effective ways to cool them at the initial development stages. It is known that in order to study effective cooling methods, it is necessary to accurately predict temperatures using CFD (Computational Fluid Dynamics). In this paper, we report the development of a temperature prediction method using thermal-fluid simulation coupled with 1D-CFD and 3D-CFD.

Key Words: *CFD, Co-simulation, Hydraulic excavator, Heat damage, Thermal fluid, Engine, Aftertreatment device, Cooling*

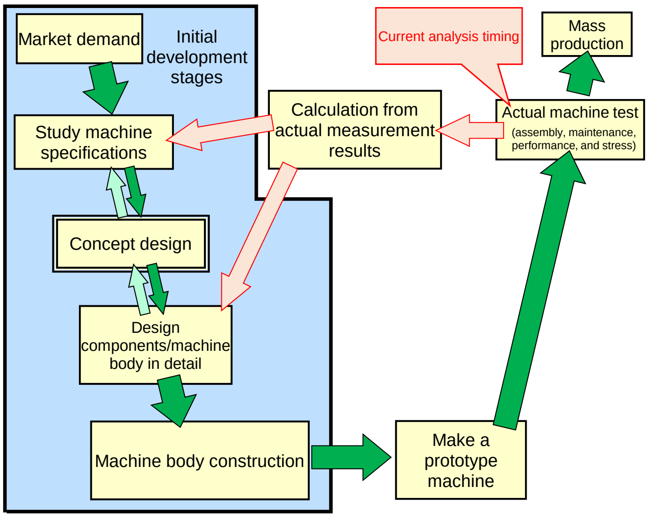
1. Introduction

In recent years, to reduce the load that construction machinery puts on the atmospheric environment, the spread of construction machinery with good emission performance has been promoted, exhaust gas countermeasures have been implemented, and emission control regulations have been tightened worldwide. Exhaust gas control systems and exhaust gas recirculation devices may be used for emission control. In that case, it is necessary to collect those high temperature devices in the engine room, and the thermal environment in the engine room becomes more and more severe.

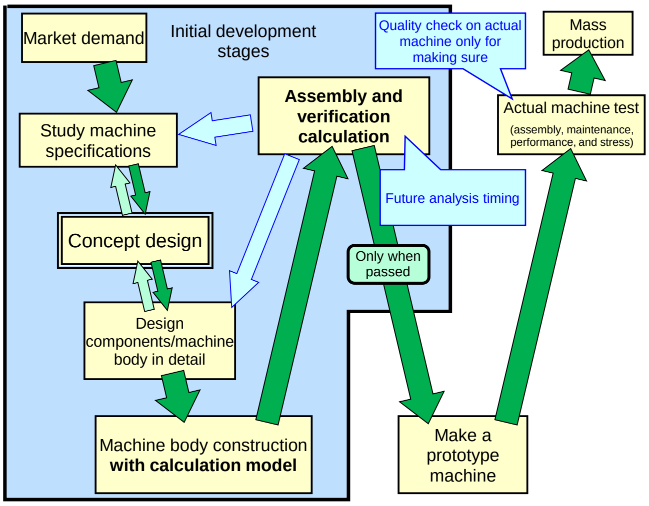
Construction machinery often operates in a fixed position under high load, and unlike automobiles, they cannot use running wind for cooling under high load. For this reason, they cool the high-temperature devices by blowing air from a high speed fan. However, since the air flow from the fan is very complicated, it is difficult to take measures against heat damage by predicting that flow. In addition, the maximum temperature of some heat-damaged parts of construction machinery can occur after the engine is stopped rather than while running under high load. Thus, we should consider the entire period from the steady state at high load to the unsteady state after the engine is stopped. The maximum temperature of the parts is, however, very difficult to study in advance for both events. A prediction method to realize front-loading of designing is desired to be developed.

Until now, heat damage was predicted by measuring the surface temperature of the heat source and performing 3D-CFD using the measured value as an input condition. This is because the surface temperature of the heat source was difficult to predict by using CFD. The surface temperature needed to be measured. This was difficult to do in the initial development stages where no actual machine was available. In addition, the heat source temperature used was a constant value obtained in the steady state, so it was difficult to check the transient change in the heat source temperature after the engine was stopped by analysis. Analyzing everything with 3D-CFD, including the heat source temperature, will require a huge amount of analysis time, which is impractical. Thus, we came up with the following: by analyzing the calorific value generated by the heat source (i.e. engine) using 1D-CFD, the heat source temperature can be predicted without actual-machine measurement and can be predicted at the initial development stages as shown in **Fig. 1**. However, we imagine that it is difficult for 1D-CFD to completely reproduce the shape of the actual parts to accurately predict the temperature distribution on the component surfaces, and the wind flow in the engine room, which is strongly affected by the structure, by calculation.

From this, we thought that by coupling 1D-CFD and 3D-CFD and reflecting the analysis results against each other, shortcomings of each method can be complemented by each other as summarized in Table 1. It will be possible to accurately reproduce the temperature conditions when the machine is in operation to predict, examine, and take measures against heat damage at the initial development stages. Believing that this would enable front-loading of designing, we proceeded with the development of the analysis method.



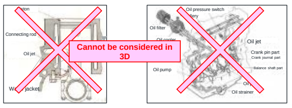
Conventional development flow



Ideal development flow

Fig. 1 Development flow

Table 1 1D-CFD vs. 3D-CFD comparison

1D-CFD	3D-CFD
• The effects of 3D structure cannot be taken into consideration. • Unsteady surrounding wind flow cannot be predicted. **⇒ Can be obtained by 3D-CFD.**	• The effects of water and oil flowing in the piping cannot be taken into consideration. • The heat source temperature needs to be measured. **⇒ Can be obtained by 1D-CFD.**
Heat transfer coefficient 40 to 200 [W/Km²] **Fluid temperature 50 to 200 [°C]**  Cannot be considered in 1D	**Heat inside the cylinder** **Engine oil flow**  Cannot be considered in 3D

2. Overview of prediction method

To make this a prediction method available for measures and studies for heat damage in the initial development stages, we constructed a model with the goal of not requiring any actual-machine measurement and completing everything with CFD. It takes a huge amount of resources and time to solve everything from the state of engine combustion to the flow in each piping with 3D-CFD, which is impractical. Therefore, we constructed a method to couple 3D-CFD with 1D-CFD, which has less computational load.

The CFD model we constructed this time consists of three models: a 1D internal fluid circuit model, a machine body model for 3D heat transfer analysis, and a machine body model for 3D fluid analysis. The 1D-CFD model solves the temperature and others of engine combustion, cooling circuit, exhaust gas, etc. The 3D-CFD models solve heat conduction in solid parts, heat dissipation to the outside, and ambient wind flow. Regarding the boundary conditions between the analysis models, as shown in **Fig. 2**, the wall surface temperature, fluid temperature, and heat transfer coefficient are transferred between the analysis models, and as shown in **Fig. 3**, coupled analysis is achieved by transferring them at regular intervals.

Figure 4 is a schematic diagram of the aftertreatment devices in the internal fluid circuit model of 1D-CFD. The granularity will be coarse as shown therein. By contrast, the machine body model for 3D heat transfer analysis reproduces the actual shape as shown in **Fig. 5**, resulting in finer granularity. Thus, coupling between 1D-CFD and 3D-CFD was achieved by dividing the machine body model for 3D heat transfer analysis according to the 1D internal fluid circuit model, transferring the averaged information, and making the granularity identical.

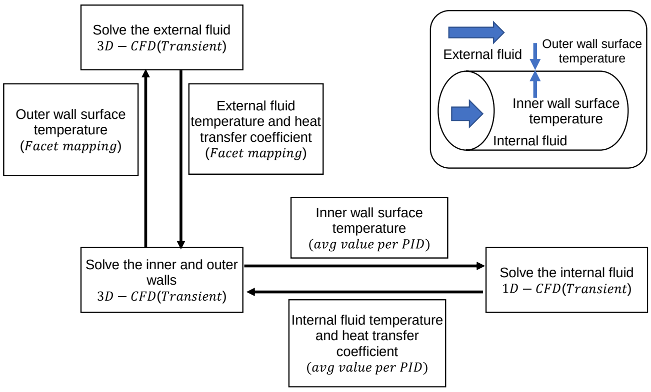


Fig. 2 Boundary conditions between the analysis models

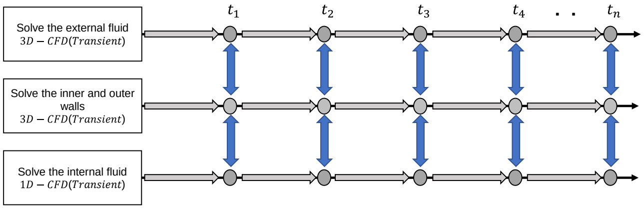


Fig. 3 Schematic diagram of data transfer between models

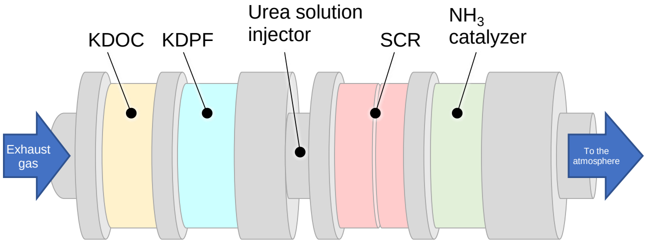


Fig. 4 Schematic diagram of aftertreatment devices in 1D model

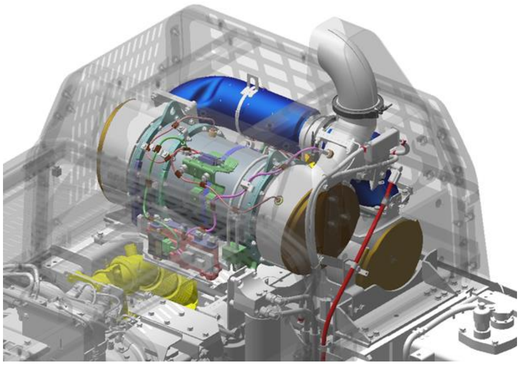


Fig. 5 Schematic diagram of aftertreatment devices in 3D model

3. Analysis model

3.1 Overview of 1D model

For this method, we constructed the model using GT-SUITE of IDAJ Co., LTD. as a 1D tool. GT-SUITE, a multiphysics system simulation tool developed by Gamma Technologies, Inc. can evaluate energy efficiency and thermal management of a cooling system in arbitrary operating conditions.

Figure 6 shows the 1D internal fluid circuit model we created this time. This model predicts the internal heat balance by modeling multiple circuits of engine combustion, cooling water, fuel, exhaust gas, etc. in order to accurately predict the surface temperature of the main heat source of the engine, aftertreatment devices, etc.

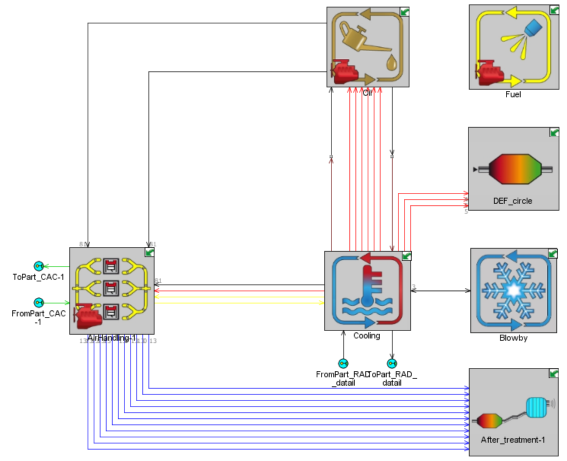


Fig. 6 Schematic diagram of 1D internal fluid circuit

3.2 Overview of 3D model

For this method, we constructed a model using PowerFLOW and PowerTHERM of Dassault Systèmes K.K. as 3D tools. PowerFLOW utilizes unsteady physical analysis based on the lattice Boltzmann method to accurately predict behavior in real-world conditions. PowerTHERM can predict the surface temperature and the heat flux generated by heat radiation, heat transfer, and heat convection. The combination of PowerFLOW and PowerTHERM enables achieving a wide range of heat- and fluid-related analyzes.

This time, we created two models: a machine body model for 3D fluid analysis and a machine body model for 3D heat transfer analysis.

We created the machine body model for 3D fluid analysis using the CAD model of the machine body as shown in **Fig. 7**. To reproduce the fan-formed flow field accurately, the 3D model of the fan was rotated at an arbitrary rotation speed in the analysis. The heat exchanger has also been modeled to accurately predict the ambient temperature in the engine room. The temperature change in the heat exchanger was solved by inputting the values of the heat input from the engine, the internal flow rate, and the performance of the heat exchanger. In an actual machine body, warm air in the engine room may wrap around in front of the heat exchanger through openings and gaps. The ambient temperature in front of the heat exchanger has a large effect on each performance and the temperature of each part in the engine room; thus, phenomenon analysis is enabled by faithfully reproducing the shape. As for the space mesh, as shown in **Fig. 8**, the finest mesh is placed in that area near the fan, which has a high wind velocity and has the greatest effect on analysis accuracy.

As shown in **Fig. 9**, in the machine body model for 3D heat transfer analysis, only the engine room outer covering and the components, rather than the entire machine body, were modeled on a per-part basis to reduce the calculation cost. The work equipment, tracks, cab, etc. were not modeled. The insulation in each section was modeled to improve the accuracy. For the piping, machine covers, insulation, etc., only the surface shape was modeled, and the calculation cost was further reduced by giving a virtual thickness.

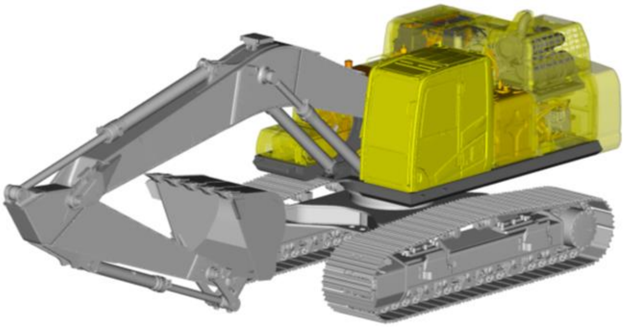


Fig. 7 Machine body model for 3D fluid analysis

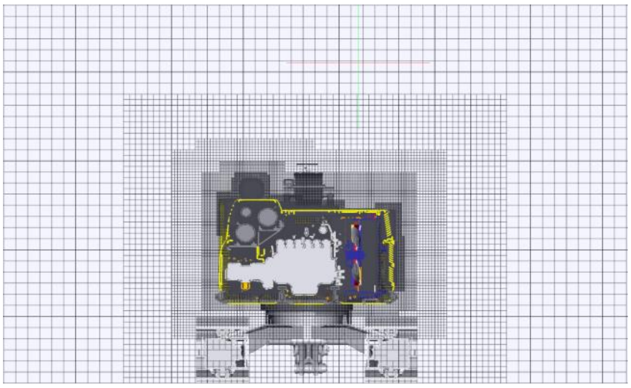


Fig. 8 3D fluid analysis space mesh (fan shaft cross-section)

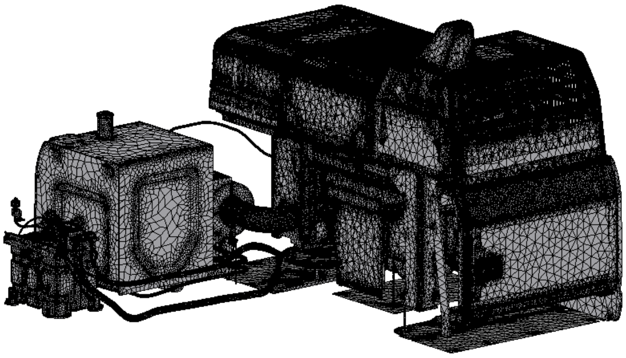


Fig. 9 Machine body model for 3D heat transfer analysis

3.3 Method of coupled analysis

This method uses the existing mechanisms of PowerFLOW and PowerTHERM to achieve data transfer between the machine body model for 3D fluid analysis and the machine body model for 3D heat transfer analysis. Data transfer between the 1D internal fluid circuit model and the machine body model for 3D heat transfer analysis was achieved by combining the function to connect with external tools of GT-SUITE and a newly created script for PowerTHERM (because there was no existing mechanism to connect GT-SUITE and PowerTHERM). This script is designed to transfer data for each part by associating the 1D internal fluid circuit model with the machine body model for 3D heat transfer analysis in advance.

This method can accurately predict the phenomenon by solving all the analyzes unsteadily; however, compared to the other two analyzes, the analysis with the machine body model for 3D fluid analysis requires a huge amount of computational resources and time. On the other hand, looking at the change in the phenomenon, the wind flow changes in a very short time compared to the change in heat, and it stabilizes quickly if no disturbance occurs. Therefore, we performed coupling while the fluctuation of the wind flow was large, but referred to a fixed flow field instead of calculating while the wind flow was stable. We were able to reduce the calculation cost and complete the analysis in a reasonable amount of time.

Figure 10 shows a schematic diagram of the coupling timing described in the previous paragraph. The **figure** shows the entire analysis procedure as well as the changes in the air volume and temperature from when the machine body is in operation to after the engine is stopped. The blue curve shows the change in air volume in the engine room, and the red curve shows that in the temperature of the parts. The blue arrows indicate the period during which it is performing 3D fluid analysis, and the orange arrows indicate that during which it is performing 1D analysis and 3D heat transfer analysis. In ranges (1) and (3) the fluctuation of wind flow is large, and in ranges (2) and (4) the wind flow is stable.

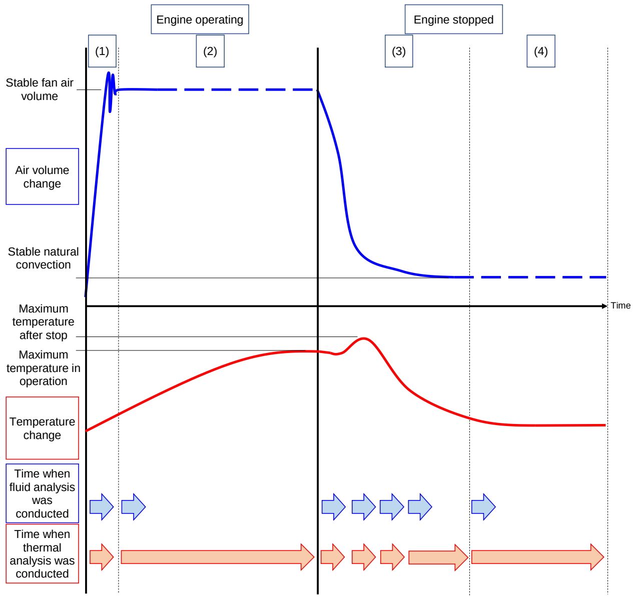


Fig. 10 Schematic diagram of analysis procedure

4. Component temperature prediction and accuracy verification results

In the study in this section, we conducted an actual work test on a hydraulic excavator, and (by using a thermocouples as shown in **Fig. 11**) measured the temperature of each part and the heat source surface temperature when the machine body was operating and after the engine was stopped. The measurement points included the surface of each part in the engine room and the high-temperature heat source parts of the engine and aftertreatment devices. Detailed measured values were obtained by conducting multi-point measurement for large parts exposed to different wind contacts depending on the location (e.g. engines and aftertreatment devices).

The actual work test on the hydraulic excavator conducted in this verification repeated the following cycle for a long time as shown in **Fig. 12**: Dig → boom RAISE & swing → dump → boom LOWER & swing → dig. **Figure 13** shows the results of verifying the effects of swinging motion in advance by analysis. Based on the calculated average temperature for one cycle, the change in the outside ambient temperature in front of the heat exchanger was approx. +1.5°C. Therefore, we judged that the effect on the surface temperature was small, and conducted the coupled analysis without swinging motion. The temperature rose during the first swing and decreased during the second swing. This is due to the design of the machine body that during the first swing, the intake opening moved toward the warm air coming out of the exhaust opening, and during the second swing, the intake opening moved away from the warm air.

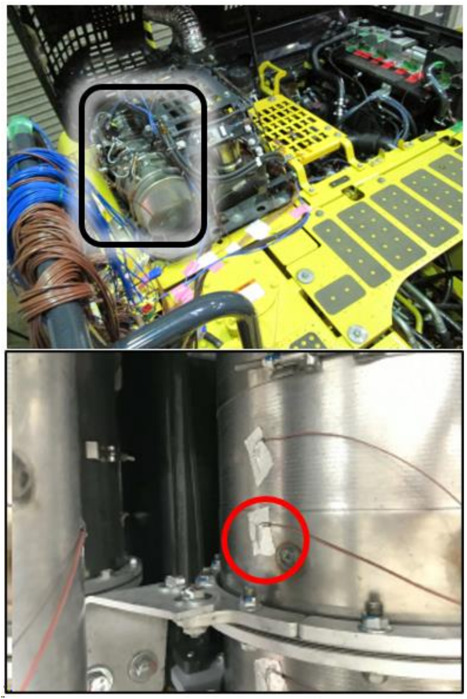


Fig. 11 Scenes of actual-machine measurement (Bottom: Enlarged view of an aftertreatment device surface with a thermocouple in the red circle)

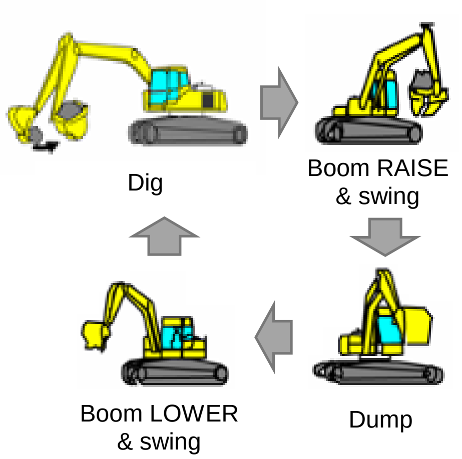


Fig. 12 Schematic diagram of actual work test

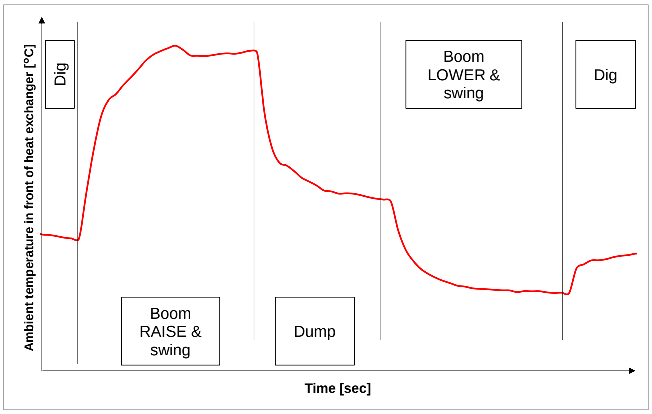


Fig. 13 Change in temperature in front of heat exchanger due to swinging motion

4.1 Prediction results of component temperature in the engine operating state

We performed comparison verification of the maximum temperature of each part in the engine operating state. **Figure 14** shows the distribution of the ambient temperature in the engine room, and **Fig. 15** shows the distribution of the engine and aftertreatment device surface temperatures. These clarify that the exhaust manifold and the aftertreatment devices were extremely hot and the ambient temperature was almost uniform in the engine room.

Figure 16 shows the surface temperatures of the parts to be verified in the engine room: it includes the results of testing and analysis at a total of 32 measurement points. We were able to predict the surface temperature of every part in the engine operating state within an error of ±10°C.

The following are the results of verifying the surface temperature of the main heat sources. **Figure 17** shows the results of the surface temperature of the aftertreatment devices, which is one of the main heat sources: it includes the results of testing and analysis at a total of 32 measurement points. The maximum error in the aftertreatment device surface temperature in the engine operating state was -23°C, and we were able to predict the temperature within an error of ±10°C for 85% of the measurement points.

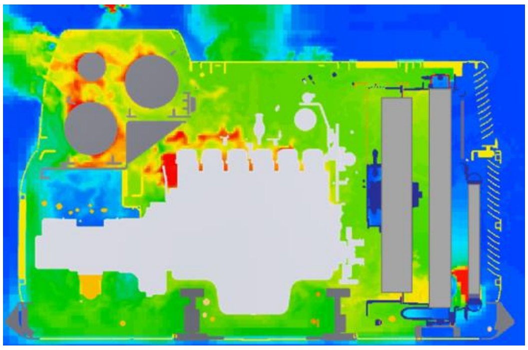


Fig. 14 Ambient temperature distribution in the engine operating state

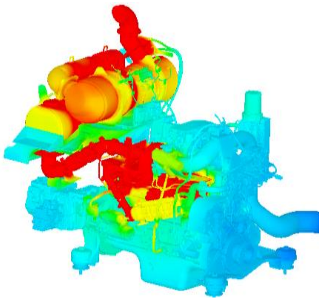


Fig. 15 Surface temperature distribution in the engine operating state

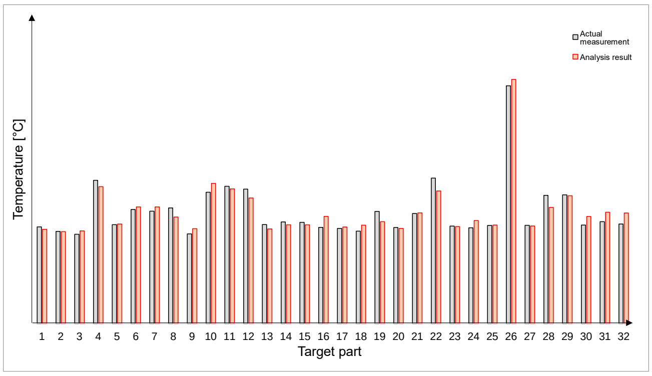


Fig. 16 Component temperatures in the engine room (measured values vs. analysis results)

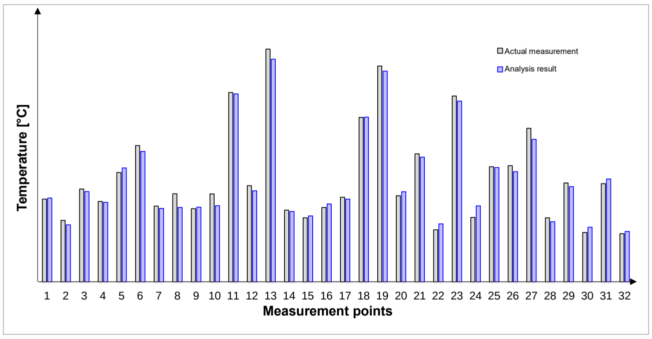


Fig. 17 Aftertreatment device surface temperatures (measured values vs. analysis results)

4.2 Prediction results of component temperatures after the engine is stopped

We performed comparison verification of the maximum temperature of each part after the engine is stopped. Since the time when the temperature reached the maximum varied depending on the measurement point, we followed the analysis results by time history. **Figure 18** shows the distribution of the ambient temperature in the natural convection state where the air volume was stable in the engine room. It reveals that hot air stayed in the engine room after the stop, unlike in the operating state.

Figure 19 shows the change over time in that part temperature after the engine is stopped that was obtained by sensors attached to the aftertreatment devices. The figure includes a solid curved line representing the measured values and dashed curved lines representing the analysis results. We were able to predict the maximum surface temperature of each part after the engine is stopped within an error of ±10°C. In addition, we confirmed that there was a very good agreement between the time when the maximum temperature was reached and how the temperature rose.

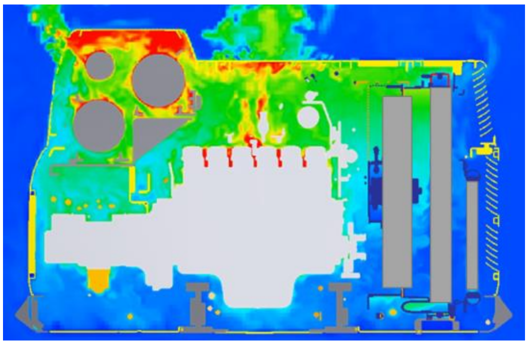


Fig. 18 Contour plot of the ambient temperature after the engine is stopped

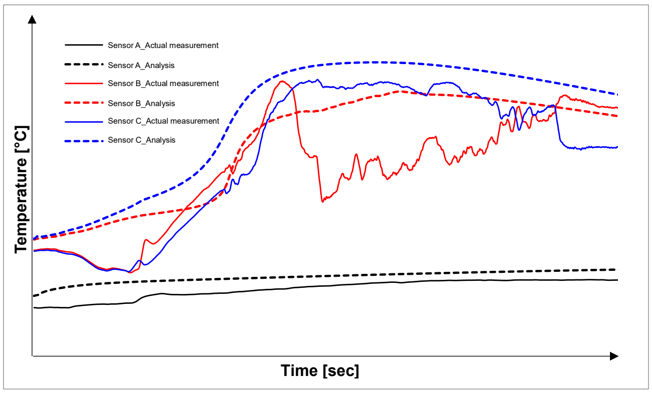


Fig. 19 Target part temperature (measured values vs. analysis results)

5. Conclusion

The R&D in this report "Development of Temperature Prediction Method for Components in Engine Room of Construction Machinery Using Thermal-fluid Simulation Coupled with 1D-CFD and 3D-CFD," aimed to develop an analysis method to enable prediction, examination, and countermeasures for heat damage at the initial development stages and achieved it by accurately reproducing the temperature situation when the machine is in operation. We believe that front-loading of designing can be realized by using this method. The following summarizes the results of the R&D:

- We have succeeded in constructing a temperature prediction method that couples 1D-CFD and 3D-CFD and does not require actual measurement of heat source surface temperatures.
- The heat source surface temperature could be predicted within an error of ±10°C for 85% of the verified measurement points.
- The maximum temperature of parts in the engine room could be predicted within an error of ±10°C at 100% of the verified measurement points.
- We could also predict temperature changes over time (e.g. after the engine is stopped).

Acknowledgments

We would like to take this opportunity to express our deep gratitude to all of IDAJ Co., LTD. and Dassault Systèmes K.K. for their great cooperation and advice in the development of this method. We look forward to your continued support for the further development of the coupling method.

References

[1] Tanaka, Y., et al., "Development of Thermal Soak Analysis on Under Hood Heat Damage after Engine Shut Off," Automotive Technology Papers Vol. 44, pp.703-708, 2013.

[2] Wang, Z., et al., "Prediction Method for Temperature Change of Parts after Vehicle Key Off," Automotive Technology Papers Vol. 46, pp.173-178, 2015.

[3] Ishikawa, K., et al., "Development of Vehicle Thermal Management Simulation," Automotive Technology Papers Vol. 50, pp.148-153, 2019.

[4] Ishikawa, K., et al., "Prediction of Component Temperatures in Cooling Fan Operating State after Key-off," Vol. 51, pp.473-478, 2020.

Introduction of the authors

Masaya Yuguchi
Joined Komatsu Ltd. in 2017.
Advanced and Core Technology
Center, Development Division

Yoshiaki Yoshida
Joined Komatsu Ltd. in 2007.
Advanced and Core Technology
Center, Development Division

Sho Watanabe
Joined Komatsu Ltd. in 2013.
Advanced and Core Technology
Center, Development Division

Takaaki Mizokami
Joined Komatsu Ltd. in 2016.
Advanced and Core Technology
Center, Development Division

[A comment from the authors]

We think that this method is very effective in the initial development stages because it does not require measurement. A high skill is required to create a model, but if this method can be utilized company-wide, it will not only improve the efficiency of development but will also lead to further improvement of product appeal by increasing the study time. In recent years, the use of IoT, machine learning, AI, etc. has made progress in improving accuracy, facilitation, automation, and optimization, and if this method can be combined with such technologies, further technological improvement can be expected.